# FINITE ELEMENTS ANALYSIS OF STRESS PEAKS IN IMPLANTS WITH DIFFERENT LENGTHS

[1]Camilo Andrés Villabona López, [2]Marco Antônio Amorin Vasco, [3]Iara Augusta Orsi, [4]Carla Cecilia Alandia-Róman, [5]Antônio Carlos Cardoso, [6]Osvaldo Luis Bezzon.

[1]Master in Oral Rehabilitation-University of São Paulo, Ribeirao Preto, Brazil; Post Graduate Doctorate in Implantology-Federal University of Santa Cataria, Florianópolis
[2]PHd in Oral Rehabilitation-University of São Paulo, Ribeirão Preto, São Paulo, Brazil; Post Doctorate in Bioengineering at the University of Zaragoza-Spain.
[3]Full Professor -School of Dentistry, Department of Dental Materials and Prosthodontics - University of São Paulo, Ribeirão Preto, São Paulo, Brasil.
[4]Research coordinator – FACIT, Faculty of Science of Tocantins. PHd in Oral Rehabilitation-University of São Paulo, Ribeirão Preto, Brazil.
[5]Full Professor -School of Dentistry - Federal University of Santa Catarina, Santa Catarina, Brazil.
[6]Full Professor -School of Dentistry- University of São Paulo, Ribeirão Preto, Brazil.

*Autor responsable de correspondencia: Camilo Andrés Villabona López*
*Correo electrónico: lacho.villa@hotmail.com*

**RESUMEN**

***Objetivo:*** este estudio se propuso evaluar, mediante el método de análisis de los elementos finitos, los valores de pico de tensión en los implantes con diferentes longitudes sometidos a carga axial y oblicua.

***Métodos y materiales:*** a partir de una simulación virtual en tercera dimensión se modeló un implante de conexión hexagonal externa con longitudes de 5,5 mm, 10 mm, 18 mm. El ensayo fue realizado bajo aplicación de carga axial y oblicua de 60 N en la superficie oclusal de la corona protética - proveniente de una reconstrucción virtual de tomografía computarizada. La simulación fue analizada numérica y cualitativamente de acuerdo con las tensiones principales (tracción-compresión).

***Resultados:*** los resultados mostraron que la carga oblicua induce mayores picos de tensión en comparación con la carga axial. Las tensiones se localizaron en las primeras roscas del implante y región periimplantar. El implante de menor longitud presentó mayor pico de tensión (100%). Los valores de pico de tensión disminuyen en promedio un 28,3% a medida que aumenta la longitud del implante de 5 a 18 mm y el 7,75% de 10 a 18 mm.

***Conclusión:*** aunque los valores de pico de tensión disminuyen a medida que aumenta la longitud de 5,5 mm a 18mm; de 10 a 18mm los valores no tienen influencia porcentual en los picos de tensión. [Villabona CA, Vasco MA, Orsi IA, Alandia-Róman CC, Cardoso AC, Bezzon OL. Finite elements analysis of stress peaks in implants with different lengths. Ustasalud 2015;14:13-18].

**Palabras clave:** análisis del elemento finito, prótesis e implantes, ingeniería.

**ABSTRACT**

***Aim:*** the purpose of this study was to evaluate, through the finite elements method, the stress peak values in implants submitted to axial and oblique loads.

***Methods:*** from a three-dimensional virtual simulation, an external hexagonal connection implant was modeled with 5.5 mm, 10 mm and 18 mm lengths. The test was conducted under application of axial and oblique loads of 60 N on the occlusal surface of the prosthetic crown from a virtual reconstruction of computed tomography. The simulation was analyzed numerically and qualitatively according to the principal stresses (tensile-compression).

***Results:*** the results showed that the oblique load induces higher stress peaks compared to the axial load. Stresses were located in the first threads of the implant and the peri-implant region. The implant of smaller length presented the highest stress peak. The stress peak values decreased an average of 28.3% with the average increase in implant length from 5 to 18 mm, and 7.75% from 10 to 18 mm.

***Conclusion:*** although the stress peak values decreased with the increase of the length from 5.5 mm to 18mm, the variations from 10 to 18mm did not influenced the stress peaks.

**Keywords:** Finite element analysis, prostheses and implants, engineering.

*Recibido para publicación: 18 de febrero de 2015. Aprobado para publicación: 8 de abril de 2015.*

## INTRODUCTION

Peri-implant bone loss, loosening and fracture of the abutment screw and prosthesis are clinical problems related to the failure of an implant-supported rehabilitation associated with bad hygiene habits and systemic factors that biologically compromise health[1].

The Finite Element Method (FEM) helps to understand the mechanical behavior of implant structures on oral rehabilitation; evaluate how a force directly influences the material of the prosthesis, prosthetic screw, abutment screw, abutment, implant, and peri-implant región[2].

In other words, FEM is a technique that aims to obtain solutions to complex mechanical problems. It is a tool which determines the stresses of an object by mathematical analysis which is subdivided into connected components in nodal points described by differential equations obtained in different dimensions[3-5]. Therefore, FEM is suitable research option to evaluate the resulting stresses of the external force, pressure, thermal changes and other physical-mechanical behavior responses[6-8].

The main complication in studies of aesthetic rehabilitation with implants is the loss of marginal bone crest. There are biological factors such as bone quality and quantity, oral hygiene, plaque and mechanical factors such as type of load, design and prosthetic material properties, prosthetic component, connection type, abutment and implant positions, diameter and length[9-16]. The aim of the present study is to evaluate the impact and influence of different implant lengths on peri-implant tensions.

## MATERIAL AND METHODS

To perform the tests by the analytical method of finite elements, a computer-aided design software (Ansys Design Modeler v10, Ansys, Canonsburg, PA, USA) was initially used to construct the three-dimensional models of the study. The model consisted of a cylindrical threaded implant of external hexagon connection with a 4.1 diameter, 4.1 mm platform and 5.5, 10 and 18 mm lengths, an universal abutment with external hexagon type connection, 4.1 mm lower platform and tapered upper portion and a titanium bolt screw with threads only in the lower third.

For the digital model of the prosthetic crown, a CT scan was initially performed covering the mandible region in cross-sections of 0.25 mm distance for a total of 212 slices. These were recorded in DICOM (digital imaging and communications in medicine standard) and imported to software "InVesalius 3.0" image processing program (Renato Archer center of technological information, Campinas, SO, Brazil) to subsequently obtain the digital reconstruction of the mandible in a 3D model (Figure 1). After the virtual reconstruction, the crown model was exported to Ansys Design Modeler.

Only element 35 was used to provide the shape and dimension of the definitive prosthetic crown (Figure 2). The prosthetic crown was modeled and edited into two parts: a crown with feldspathic porcelain covering the infrastructure in chrome cobalt with at least 0.3 mm thick. Between the crown and the implant abutment, an approximate thickness of 0.1 mm was modeled simulating the zinc phosphate cement to analyze with finite elements the cement layer. The cortical bone thickness of 1.0 mm covering the medullary bone (Figure 3) and three enamel cylinders were distributed on the occlusal surface of the prosthetic crown to simulate occlusal contacts (Figure 4).

Three models with implants of 5.5 mm, 10 mm and 18 mm lengths were exported to ANSYS Workbench v10 finite elements software (Ansys, Canonsburg, PA, USA). Except for the length and size of the channels in the apical region of the implant, all models were identical (Figure 5). To submit the models to forces simulating the masticatory load, rigid supports in the lower and lateral regions of the bone were added, simulating the union of the model to a mandible; axial and oblique loads of 60 N intensity only in the central contact were simulated in the prosthetic crown 45º.

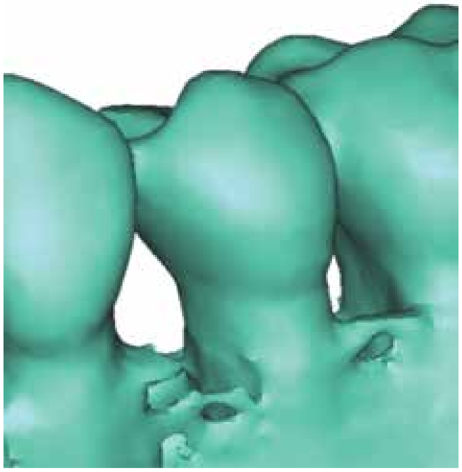

**Figure 1.** Digital reconstruction from CT without any editing.

**Figure 2.** Editing of dental element 35.

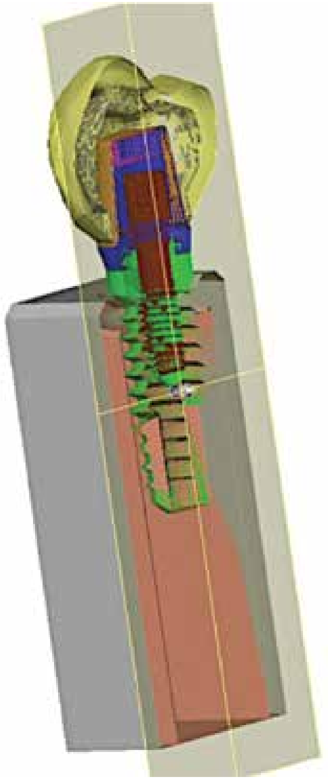

**Figure 3.** Figure of final cross-sectional model: medullary bone (pink), cortical bone (gray), implant (green), passante screw (brown), Trunnion (blue), guttapercha (Pink), zinc phosphate cement ( light brown) infra metal frame (light gray) and ceramics (yellow).

**Figure 4.** Brown enamel cylinders simulating occlusion contacts.

**Figure 5.** View of the implant lengths: model a (5.5 mm), model b (10 mm) and c (18mm).

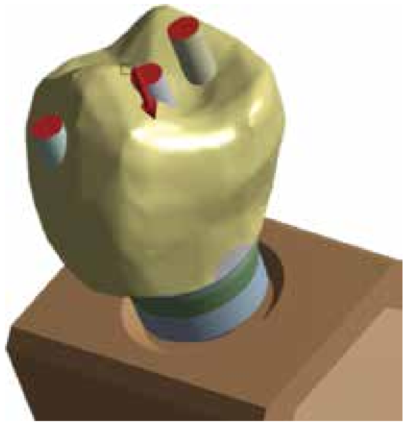

**Figure 6.** Areas (red) of application loads and direction vector (arrow).

Relative to the long axis of the implant (Figure 6). The structures were configured in relation to the mechanical properties according to their elastic modulus and Poisson's ratio (Table 1).

**Table 1.** Mechanical properties of the materials

| Materials | Young Modulus (MPa) | Poisson Ratio (MPa) |
|---|---|---|
| Cortical bone[17] | 13700 | 0,3 |
| Medullar bone[17] | 1370 | 0,3 |
| Zinc phosphate cement[17] | 22400 | 0,25 |
| Guta percha[17]) | 0.69 | 0,45 |
| Enamel[18] | 84100 | 0,20 |
| Feldspatic porcelain[19] | 69000 | 0,3 |
| Titanium[20] | 110000 | 0.35 |
| Crome-cobalt structure[21]) | 218000 | 0,33 |

(Holmes et al.17, Menicucci et al.18, Zarone et al.19, Benzing et al.20, Eskitascioglu et al.21).

The discretization was carried out, that is, the transformation process of the model in nodes and elements required for the simulation. The meshes called "finite elements", were generated and validated by a refining process, as the number of nodes and elements were gradually increased until the difference in stress peaks between a mesh refinement and another was 3% or less. Thus, the geometric error characteristic of a mesh discretization process was minimized. The mesh was generated using 10 node quadratic tetrahedral elements (solid 187), enabling an appropriate copy of irregular geometries (Figure 7). The generated meshes had similar configurations and varied from 1,194,990 nodes and 732,317 elements to 1,060,094 nodes and 654,542 elements. All the models were them solved (Windows XP x64, Intel Core 2 Quad Q6600 processor, 8 GB RAM). The graphical and numerical plots of data were recorded, evaluated and compared thorugh qualitative and quantitative analysis.

**Figure 7.** Mesh of the finite element model.

The quantitative results (MPa) were automatically generated by the ANSYS software and, for the qualitative analysis, the results were converted into percentage values (%) and, in order to compare the models, the highest MPa result was considered as a 100% value.

## RESULTS

Two types of analyses were performed. The first was the qualitative analysis, which verified the distribution and sites of greatest stress intensity. The graphical plot of results for the axial load (Figure 8 a and b) showed that the maximum values of traction and compression were located at the cervico-apical area of the 5mm screw (Table 2). For the oblique load (Figure 9 a and b), the peak stress for both traction and compression was located at the cervical region corresponding to the first three threads of the 5.5mm screw.

The second analyses was the quantitative analysis, which verified and compared the numerical intensity (in percentage) of the stress peaks of the peri-implant region. For the axial load (Table 2) the peak stress decreased on average 30.5% and 44.5% when the lengths were increased to 10 mm and 18 mm, respectively. When the length was increased from 10mm to 18mm, the stresses decreased 14%. With regard to oblique loading (Table 3), the stress decreased on average 18.5% and 20% when the lengths were increased from 10 mm to 18 mm, respectively, and 1.5% from 10 mm to 18 mm. The stress peak values decreased on average 28.3% when the length of the implant was increased from 5 to 18 mm, and 7.75% from 10 to 18 mm.

**Table 2.** Results of the peri-implant stress peaks, axial load

| Lengths of implants | Tensile | Compression |
|---|---|---|
| **Model A (5,5 mm)** | 22,35 MPa (100%) | 32,01 MPa (100%) |
| **Model B (10 mm)** | 15,13 MPa (67%) | 23,34 MPa (72%) |
| **Model C (18 mm)** | 11,54 MPa (51%) | 19,41 MPa (60%) |

**Table 3.** Results of the peri-implant stress peaks, oblique load

| Lengths of implants | Tensile | Compression |
|---|---|---|
| **Model A (5,5 mm)** | 127,94 MPa (100%) | 145,3 MPa (100%) |
| **Model D (10 mm)** | 106,81 MPa (83%) | 117,56 MPa (80%) |
| **Model H (18 mm)** | 106,65 MPa (83%) | 112,38 MPa (77%) |

## DISCUSSION

According to the results obtained, the oblique load induces higher stress peaks compared to axial load, making it more harmful to the peri-implant health. Similarly, Eskitascioglu et al. (2004)[21] reported that the moments that cause the greater stress load is not the axial load but the oblique load, since an oblique load represents a more realistic occlusal direction due to the mandibular movements, premature and/or high contacts. Although non-axial load on implants has been documented as the most pathological one, due to the risk of stress concentration exceeding the physiological support capacity of cortical bone [2], in the literature, there is no consensus regarding which load would have the ability to cause higher stress and lead to bone loss (marginal ridge) and/or prosthetic components fracture. There are also no exact stress values that can lead to bone loss due to variations in mechanical (length, diameter, position, the implant surface and prosthetic components) and biological factors (bone quality, occlusal force and health conditions of the patient) from patient to patient[22-26].

The results of this study showed that the stress peak values decreased with increasing length; does it mean that the largest length has the greatest success? It is so according to Winkler, Winkler et al., (2000)[27] who performed a clinical trial for three years of follow up in 2,900 implants with 7 mm, 8 mm, 10 mm, 13 mm and 16 mm lengths and 3.0 and 4.0 mm in diameter positioned in the maxilla. The researchers showed that short implants had a lower success rate than longer implants, being also confirmed by the clinical study of Morris et al., (2004)[28], who for six years, verified the success rate of over 1,500 long implants in the maxilla and mandible. The results showed that the lowest success rate (89.4%) was for implants with a length of 8 mm; for 11mm implants, it was 94.3% and 17 mm implants obtained the highest rate (97.7%). Researchers have shown that using a longer implant provides better biomechanical results and the increase in the length of the implants is related to the highest success rates of oral rehabilitation. Similarly, Renouard and Nissan (2006)[29] confirm that short implants 8 mm in length have been documented with lower success rates (78%) in rehabilitation treatments with implants and has a tendency for more frequent failures than longer implants.

Using the three-dimensional finite elements analysis, Tada et al. (2003)[26] conducted a study to assess the influence of stress on implants with different lengths, positioned in areas with different bone densities. Implants with 9.2 mm; 10.8 mm; 12.4 mm and 14.0 mm lengths were positioned in four bone types (Type I, anterior mandible region, Type II, posterior mandible region; Type III, the maxillary anterior region and Type IV, the maxillary posterior region). Axial and lingual vestibular forces were applied to the occlusal and central regions of the prosthetic abutment. The results showed that regardless of the load direction, the implants with greater lengths had better stress distribution than shorter implants, particularly when placed in bone with less density. The maximum principal stress (traction) peak increased wihen the implant length decreased. It was verified that higher bone density can ensure better biomechanical behavior, thus longer implants might be the best choice for regions with bone corresponding to the mandibular bone.

Misch et al. (2006)[30] noted that the use of short implants offers several advantages compared to longer implants, such as shorter time, cost, surgical risk, discomfort, and major surgical facilities and maintaining bone tissue. Risk factors that increase the stress peaks and failure in treatment with short implants are mainly due to the increased height of the prosthetic crown, the greater bite force and bone density. To reduce the elevated stress peaks, cantilever prostheses should be avoided and, when possible, implants should be splinted in order to improve the load distribution.

Short implants are a viable solution in cases with reduced bone height, however, one must be aware and avoid excessive occlusal height or the buccal design of the prosthesis due to the increased magnitude forces in restorations. Malo, Araujo and Rangert (2007)[31] reported that such implants can be a viable long term option for restorative treatments in the maxilla and mandible because the implant length, short (7.0 mm) and long (10.0 mm) did not significantly affect the tension gradients. The present study is in agreement with Malo, Araujo and Rangert, since on average, the increase in length of the stress peaks did not cause a considerable proportional impact.

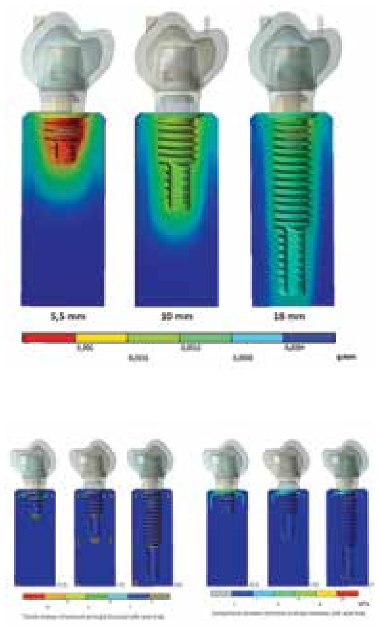

**Figure 8.** Displacement of tension with an axial load. The tension distribution in the 5.5mm screw presents maximum values in the von Mises scale compared to different lengths.

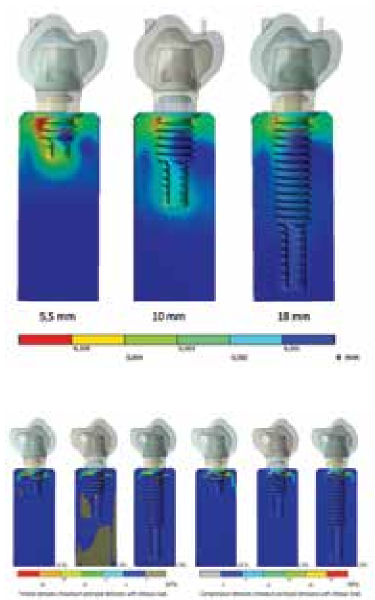

**Figure 9.** Displacement of tension with oblique loads. The tension distribution in the 5.5mm screw axis presents maximum values in the von Mises scale compared to the different implants.

## CONCLUSION

Although the stress peak values decreased with the increase of the length from 5.5 mm to 18mm, the variations from 10 to 18mm did not influenced the stress peaks.

## REFERENCES


1. Yokoyama S, Wakabayashi N, Shiota M, Ohyama T. The influence of implant location and length on stress distribution for three-unit implant-supported posterior cantilever fixed partial dentures. J Prosthet Dent. 2004;91(3):234-40.
2. Akça K, Iplikçioğlu H. Finite element stress analysis of the effect of short implant usage in place of cantilever extensions in mandibular posterior edentulism. J Oral Rehabil. 2002;29(4):350-6.
3. Clelland NL, Ismail YH, Zaki HS, Pipko D. Three-dimensional finite element stress analysis in and around the Screw-Vent implant. Int J Oral Maxillofac Implants. 1991;6(4):391-8.
4. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointegrated dental implants using finite element analysis--a two-dimensional comparative study examining the effects of implant diameter, implant shape, and load direction. J Oral Implantol. 1998;24(2):80-8.
5. Lotti, R. S. et al. Aplicabilidade científica do método dos elementos finitos. R. Dental Press. Ortodon. Ortop. Facial 2006;11:35-43.
6. Wakabayashi N, Ona M, Suzuki T, Igarashi Y. Nonlinear finite element analyses: advances and challenges in dental applications. J Dent. 2008;36(7):463-71.
7. Geng JP, Xu DW, Tan KB, Liu GR. Finite element analysis of an osseointegrated stepped screw dental implant. J Oral Implantol. 2004;30(4):223-33.
8. Huang HK, Ledley RS. Numerical experiments with a linear force-displacement tooth model. J Dent Res. 1969;48(1):32-7.
9. Byrne D, Houston F, Cleary R, Claffey N. The fit of cast and premachined implant abutments. J Prosthet Dent. 1998;80(2):184-92.
10. Binon PP. Implants and components: entering the new millennium. Int J Oral Maxillofac Implants. 2000;15(1):76-94.
11. Gross MD, Nissan J. Stress distribution around maxillary implants in anatomic photoelastic models of varying geometry. Part II. J Prosthet Dent. 2001;85(5):450-4.
12. Maeda Y, Satoh T, Sogo M. In vitro differences of stress concentrations for internal and external hex implant-abutment connections: a short communication. J Oral Rehabil. 2006;33(1):75-8.
13. Pilliar RM, Sagals G, Meguid SA, Oyonarte R, Deporter DA. Threaded versus porous-surfaced implants as anchorage units for orthodontic treatment: three-dimensional finite element analysis of peri-implant bone tissue stresses. Int J Oral Maxillofac Implants. 2006;21(6):879-89.
14. Novaes AB, Barros RR, Muglia VA, Borges GJ. Influence of interimplant distances and placement depth on papilla formation and crestal resorption: a clinical and radiographic study in dogs. J Oral Implantol. 2009;35(1):18-27.
15. De Morais Alves da Cunha, T. et al. Comparison of Fit Accuracy between Procera® Custom Abutments and Three Implant Systems. Clin. Implant Dent. Relat. Res. Dec, 2010.
16. Degidi M, Nardi D, Piattelli A. One abutment at one time: non-removal of an immediate abutment and its effect on bone healing around subcrestal tapered implants. Clin Oral Implants Res. 2011;22(11):1303-7.
17. Holmes DC, Diaz-Arnold AM, Leary JM. Influence of post dimension on stress distribution in dentin. J Prosthet Dent. 1996;75:140-7.
18. Menicucci G, Mossolov A, Mozzati M, Lorenzetti M, Preti G. Tooth-implant connection: some biomechanical aspects based on finite element analyses. Clin Oral Implants Res. 2002;13:334-41.
19. Zarone F, Sorrentino R, Apicella D, Valentino B, Ferrari M, Aversa R, et al. Evaluation of the biomechanical behavior of maxillary central incisors restored by means of endocrowns compared to a natural tooth: a 3D static linear finite elements analysis. Dent Mater. 2006;22:1035-44.
20. Benzing UR, Gall H, Weber H. Biomechanical aspects of two different implant-prosthetic concepts for edentulous maxillae. Int J Oral Maxillofac Implants. 1995;10:188-98.
21. Eskitascioglu G, Usumez A, Sevimay M, Soykan E, Unsal E. The influence of occlusal loading location on stresses transferred to implant-supported prostheses and supporting bone: A three-dimensional finite element study. J Prosthet Dent. 2004;91(2):144-50.
22. Akça K, Iplikçioğlu H. Finite element stress analysis of the influence of staggered versus straight placement of dental implants. Int J Oral Maxillofac Implants. 2001;16(5):722-30.
23. Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van der Perre G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. J Oral Rehabil. 1998;25(11):847-58.
24. Ding X, Liao SH, Zhu XH, Zhang XH, Zhang L. Effect of diameter and length on stress distribution of the alveolar crest around immediate loading implants. Clin Implant Dent Relat Res. 2009;11(4):279-87.
25. Geng, J. P.; Tan, K. B.; Liu, G. R. Application of finite element analysis in implant dentistry: a review of the literature. J. Prosthet. Dent. 2001;85:585-98.
26. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2003;18(3):357-68.
27. Winkler S, Morris HF, Ochi S. Implant survival to 36 months as related to length and diameter. Ann Periodontol. 2000;5(1):22-31.
28. Morris HF, Ochi S, Crum P, Orenstein IH, Winkler S. AICRG. Part I: A 6-year multicentered, multidisciplinary clinical study of a new and innovative implant design. J Oral Implantol. 2004;30(3):125-33.
29. Renouard F, Nisand D. Impact of implant length and diameter on survival rates. Clin Oral Implants Res. 2006;17(Suppl 2):35-51.
30. Misch CE, Steignga J, Barboza E, Misch-Dietsh F, Cianciola LJ, Kazor C. Short dental implants in posterior partial edentulism: a multicenter retrospective 6-year case series study. J Periodontol. 2006;77(8):1340-7.
31. Maló P, de Araújo Nobre M, Rangert B. Short implants placed one-stage in maxillae and mandibles: a retrospective clinical study with 1 to 9 years of follow-up. Clin Implant Dent Relat Res. 2007;9(1):15-21.


### Correo electrónico de los autores


*Camilo Andrés Villabona López: lacho.villa@hotmail.com*
*Marco Antônio Amorin Vasco: maavasco@gmail.com*
*Iara Augusta Orsi: iaraorsi@forp.usp.br*
*Carla Cecilia Alandia-Róman: ccalandia@ usp.br*
*Antônio Carlos Cardoso: antonio.cardoso@ufsc.br*
*Osvaldo Luis Bezzon: olbezzon@ forp.usp.br*