COB-2021-0456
PARAMETRIC ANALYSIS OF A UNITARY DENTAL PROSTHESIS OVER-IMPLANT USING FINITE ELEMENT MODELS
26TH COBEM

Andrade, Matheus Prevelato
Hernandez, Bruno Agostinho
Togashi, Marilia Magro
Capello Sousa, Edson Antonio
Engineering College of Bauru, São Paulo State University – UNESP, Ave. Eng. Luiz Edmundo C. Coube, 14-01, Bauru, Brazil
matheus.prevelato@unesp.br
bruno.agostinho@unesp.br
marilia.togashi@unesp.br
edson.capello@unesp.br

Abstract. Dental prostheses over-implants have been widely used in the treatment of patients with tooth loss. Despite a high success rate, mechanical failures still occur as the implant success is directly dependant on geometrical and structural characteristics of the prosthesis-implant system. Studies about the biomechanical behaviour of dental prosthesis are thus important to avoid such failures and to ensure the structural integrity of the prosthesis and the patient's well-being. In this context, parametric analysis are frequent used to find the best geometrical parameters leading to better stress levels on the bone. This study proposed a parametric analysis of a unitary dental prosthesis over-implant using the Finite Element Method, Design of Experiments (DOE) and Response Surface Methodology (RSM). A mathematical function representing the prosthesis structural behaviour as a function of its physical parameters (implant height, abutment height, cortical bone stiffness, and applied load) was developed. Through this function, the influence of the parameters was analysed and the responses were obtained in terms of von Mises stress levels in the region of the bone-implant interface. It was possible to analyse the sensitivity of each parameter and its influence on bone stress levels: all parameters linearly influenced the response, with the applied load being the most relevant. The use of a mathematical function representing the prosthesis system resulted in a more agile process to estimate critical stress levels without the need of new models. These functions will be now used in an optimisation process of biomechanical structures.

Keywords: Parametric Analysis, Dental Prosthesis, Finite Element Method, DOE, RSM.

1. INTRODUCTION

Dental prostheses are biomechanical structures used to restore the mechanical and aesthetic functions of damaged teeth. The use of osseointegrated over-implant prostheses is a common practice in dentistry due to numerous successful cases. However, according to Franciosa (2012), long-term failures still occur. External forces are applied to the structure, generating stresses and strains in the implant and surrounding bone. Skalak (1983) stated that the success or failure of a dental prosthesis is dependent on how mechanical stresses are transferred through the implant to the bone tissue. An accurate assessment of biomechanical loadings on implants is thus imperative for a correct clinical prediction of bone response and to ensure the longevity of the entire implant system (Akça et al, 2002).

Prostheses geometry has a great influence on the load distribution on implants (Rangert et al, 1989). Dental implants placed in low bone density maxillary regions are more likely to fail than those in denser regions (Genna, 2003). To increase the survival rate of dental prostheses, the initial quality of the bone is a critical factor to be considered.

Biomechanical structures present non-linearities, such as materials with non-homogeneous mechanical properties, non-linear geometries, and irregular loading and boundary conditions, which makes analytical solutions impractical. Numerical methods, as the Finite Element Method (FEM), are widely used in biomechanics. According to Baiamonte (1996), it is possible to analyse, with good precision, the mechanical behaviour of dental prostheses using FEM.

In order to develop new and better implants, it is necessary to investigate the structural parameters affecting the mechanical behaviour of the prosthesis, such as the material properties and geometry. However, if this study is conducted through in vitro experiments, it will certainly be a time-consuming process. With the increasing processing capabilities of modern computers, it is possible to automatize analytical methods to study the influence of those parameters without the need of using in vitro samples.

One analytical method widely used to evaluate the influence of several parameters in an output is the Response Surface Methodology (RSM). RMS is a set of statistical and mathematical techniques applied in the development, improvement, and optimisation of components (Myers, 2009). The purpose of this methodology is to find a mathematical equation (the Response Surface) that correlates the independent variables to the outcomes, so that, given a new set of independent variables, an approximate answer is instantaneously obtained. RSM usually uses the Design of Experiments (DOE) method to create the response surface. The DOE is based on experimenting several combinations of parameters and measuring the responses provided. The responses, which are usually obtained from several finite element models - each one with a specific combination of parameters, are then used to create the response curve (Handbook, 2012).

This study aims to perform a parametric analysis of a single dental prosthesis over-implant, using Finite Element models, and develop a mathematical function representing the structural behaviour of this prosthesis as a function of its physical parameters: implant height, abutment height, cortical bone stiffness, and applied load. In this way, we propose the application of statistical methods to obtain a response surface that represents the implant structural behaviour and describes the most influential parameters for the response.

2. MATERIALS AND METHODS

The finite element model developed in this study, Figure 1, is based on a dental prosthesis over-implant used by Albarracín (2011) and later studied by Hernandez (2015).
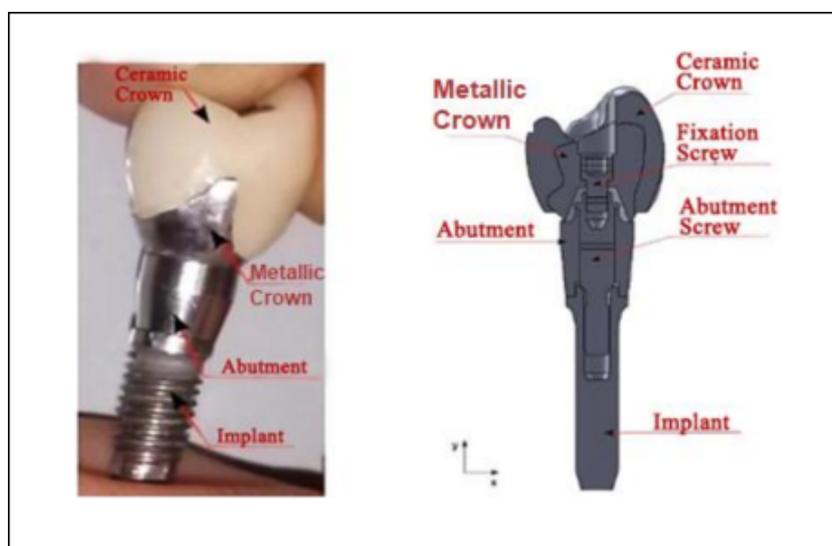

Figure 1. Real dental prosthesis (left) and dental prosthesis model (right) (Hernandez, 2015).

The prosthesis is composed of a Branemark System MK III Groove single implant (Nobel Biocare – Götembur, Sweden), with a multi-unit abutment (Nobel Biocare – Götemburg, Sweden), titanium screws, and a prosthetic crown of cobalt-chromium (Co-Cr) alloy coated with feldspathic ceramic (CNG Prosthesis Solutions, São Paulo, SP, Brazil), as shown in Figure 1.

The internal components of the prosthesis (implant, abutment, and screws) were directly modelled in the finite element software (Ansys 2019.R3, Swanson Analysis System, Houston, Pa, USA). The metallic crown, the feldspathic porcelain coating, and the mandibular bone models were previously developed by Hernandez (2015) and imported into the finite element software. The geometries of the ceramic coating and the metallic crown were based on 3D scanning and micro-CT images, respectively. The components were then assembled, Figure 2.

The mechanical properties applied to the prosthesis components were considered to be homogeneous, isotropic, and linearly elastic, Table 1.

A solid 3D quadratic element (SOLID187), with ten nodes and three degrees of freedom per node, was chosen to generate the finite element mesh. After generating the mesh, boundary conditions were applied: movement restrictions were applied at all degrees of freedom of the mandibular bone to simulate its real fixation. A load was applied at the top surface of the crown, and it ranged from 91N and 294N, Figure 3.

Contact elements were inserted in the shared surfaces between implant and abutment, and between abutment and crown. Preloads of 100N.mm and 200N.mm were applied to the implant and crown screws, respectively, to simulate the tightening torque.

A parametric analysis was conducted using Response Surface Methodology (RSM), Design of Experiments methodology (DOE), and Central Composed Face Centered (CCF) mathematical model. This model requires for three levels, or options, on each parameter: maximum value (+1), minimum value (-1), and the average value (0). Figure 4

illustrates the analysed parameters in this study: implant height (H), abutment height (h), Young's modulus of cortical bone (E), and applied load (F). Table 2 presents the real values and their equivalent in coded values for each variable.
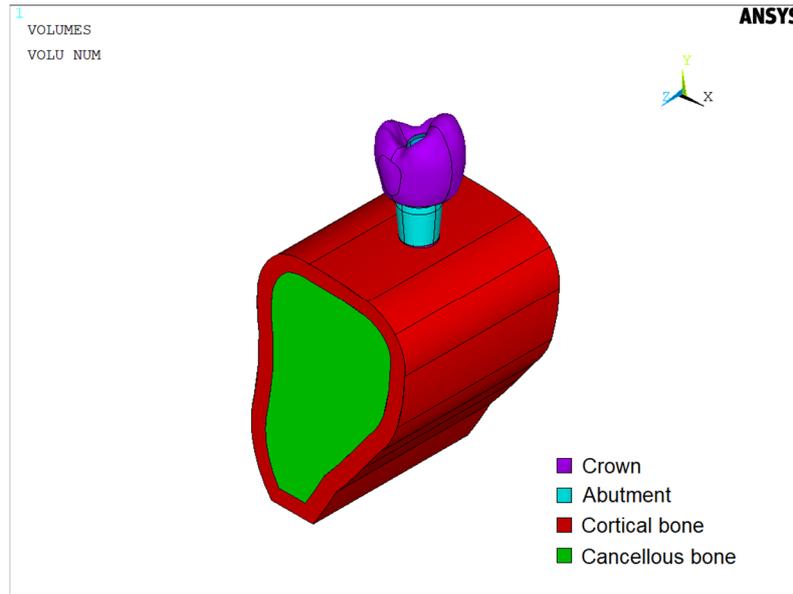

Figure 2. Final structure composed of the prosthesis incorporated into the bone.

Table 1. Mechanical properties of the components used in the model.

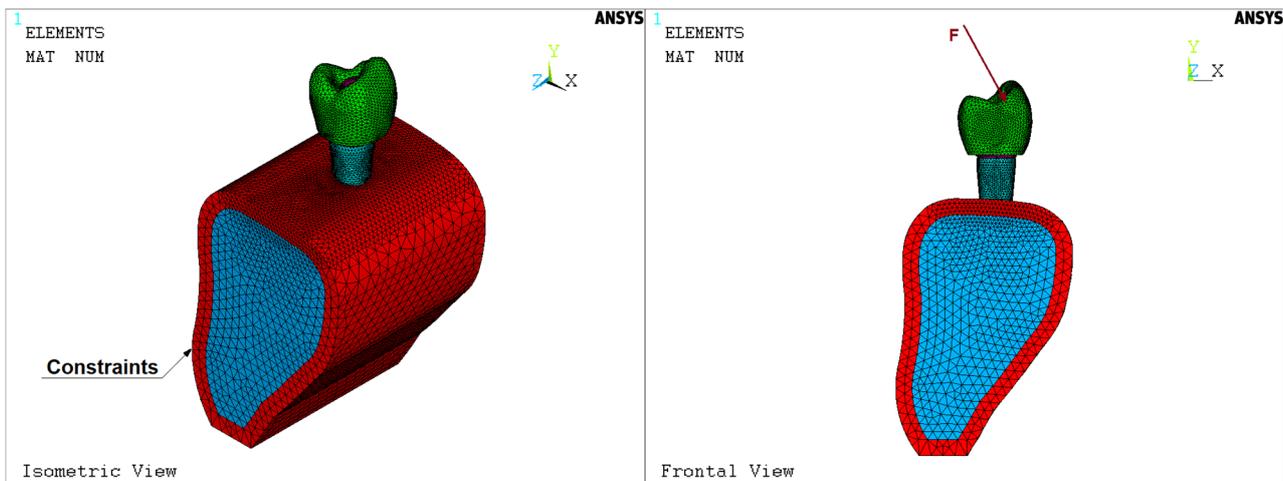| Material	Young's Modulus [GPa]	Poisson's coefficient	References
Cortical bone	14	0.3	Juodzbalys et al. (2005)
Cancellous bone	1	0.3	Juodzbalys et al. (2005)
Implant	110	0.34	Aoki et al. (2004)
Abutment	110	0.34	Aoki et al. (2004)
Fixation screw	100	0.34	Aoki et al. (2004)
Abutment screw	100	0.34	Aoki et al. (2004)
Metallic crown	218	0.33	Craig (1989)
Ceramic crown	68.9	0.28	Geng et al. (2001)

Figure 3. Discretized model (left) and load applied (right).
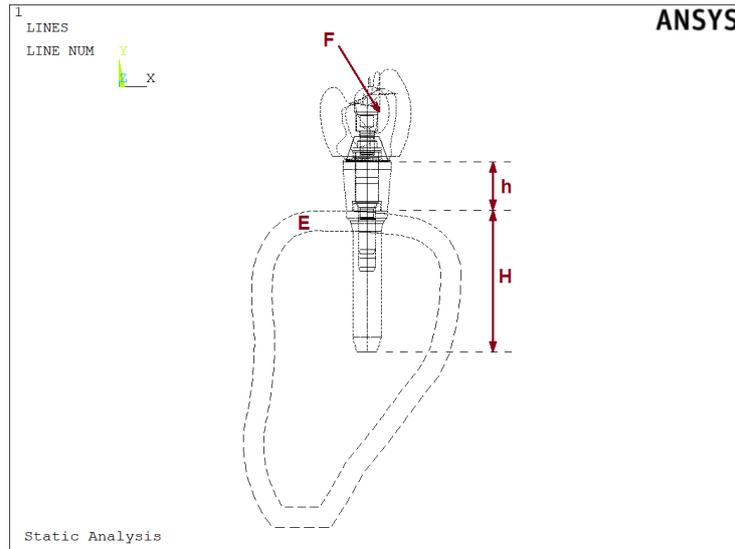
Figure 4. Analysed parameters.

Table 2. Coded and real values for each parameter.

MODEL	H_{cod}	h_{cod}	E_{cod}	F_{cod}	H_{real} [mm]	h_{real} [mm]	E_{real} [Mpa]	F_{real} [N]
1	-1	-1	-1	-1	7	1	8000	91
2	-1	-1	-1	1	7	1	8000	284
3	-1	-1	1	-1	7	1	20000	91
4	-1	-1	1	1	7	1	20000	284
5	-1	1	-1	-1	7	5	8000	91
6	-1	1	-1	1	7	5	8000	284
7	-1	1	1	-1	7	5	20000	91
8	-1	1	1	1	7	5	20000	284
9	1	-1	-1	-1	15	1	8000	91
10	1	-1	-1	1	15	1	8000	284
11	1	-1	1	-1	15	1	20000	91
12	1	-1	1	1	15	1	20000	284
13	1	1	-1	-1	15	5	8000	91
14	1	1	-1	1	15	5	8000	284
15	1	1	1	-1	15	5	20000	91
16	1	1	1	1	15	5	20000	284
17	-1	0	0	0	7	3	14000	187.5
18	1	0	0	0	15	3	14000	187.5
19	0	-1	0	0	11	1	14000	187.5
20	0	1	0	0	11	5	14000	187.5
21	0	0	-1	0	11	3	8000	187.5
22	0	0	1	0	11	3	20000	187.5
23	0	0	0	-1	11	3	14000	91
24	0	0	0	1	11	3	14000	284
25	0	0	0	0	11	3	14000	187.5

3. RESULTS AND DISCUSSION

Twenty-five (n=25) different models were generated and analysed. Their results, in terms of Von Mises stress around the implant hole on the cortical bone, were obtained as illustrated in Figure 5 and listed in Table 3.
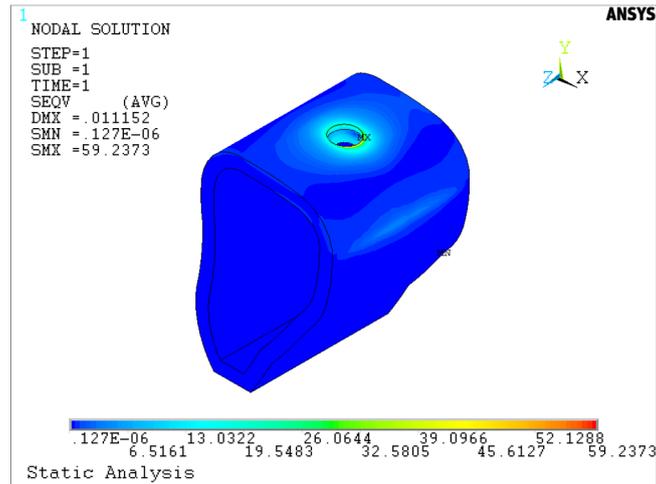
Figure 5. Von Mises stress on the cortical bone.

Table 3. Von Mises stresses in the cortical bone obtained on each analysis.

MODEL	S_{calc} [MPa]	MODEL	S_{calc} [MPa]	MODEL	S_{calc} [MPa]	MODEL	S_{calc} [MPa]	MODEL	S_{calc} [MPa]
1	17,982	6	64,692	11	19,906	16	72,350	21	29,229
2	37,763	7	27,904	12	41,707	17	42,591	22	40,387
3	24,980	8	87,085	13	16,661	18	34,852	23	15,685
4	48,304	9	13,876	14	51,994	19	27,596	24	56,550
5	20,729	10	31,886	15	23,183	20	43,840	25	36,118

Based on the Von Mises stress values, three different equations were obtained using Microsoft Office Excel software. These equations were: a linear regression, a linear regression with variable interaction, and a non-linear regression. The best result, in terms of adjusted R^2, was obtained from the linear regression with variable interaction as a function of implant height (H), abutment height (h), Young's modulus of cortical bone (E), and applied load (L). This equation is shown in Eq. (1).

$$S_{est} = 37.114 - 3.645H + 8.024h + 5.610E + 17.301F - 0.910Hh - 0.273HE - 1.371HF + 1.440hE + 6.545hF + 2.274EF \qquad (1)$$

The significance of each parameter was analysed using a p-value analysis, Figure 6. The equation terms which represent the interaction between variables, implant high and abutment high (H×h) and implant high and Young's modulus (H×E), had low significance (p-value < 0.05) and, therefore, were removed, simplifying the response surface, Eq. (2).
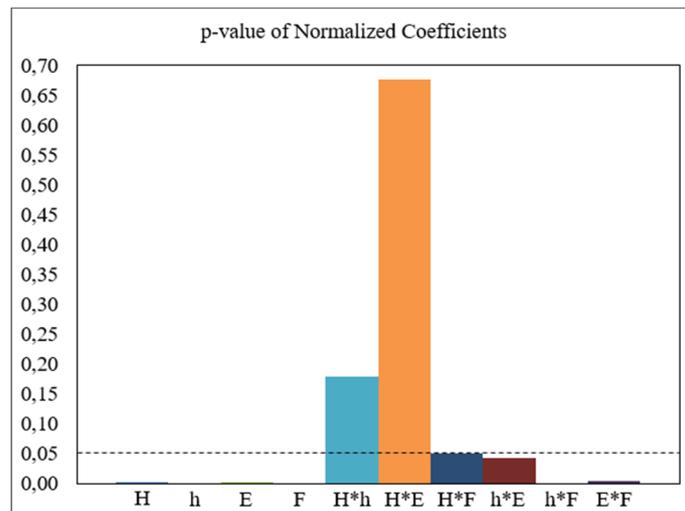

Figure 6. p-value of normalized coefficients of linear regression with interaction.

$$S_{est} = 37.114 - 3.645H + 8.024h + 5.610E + 17.301F - 1.371HF + 1.440hE + 6.545hF + 2.274EF \tag{2}$$

To evaluate the accuracy of the response surface (Eq. (2)), new structural parameters (H_{new}, h_{new}, etc) were randomly defined, and von Mises stresses were estimated by the response surface (Eq. (2)). The results were then compared to von Mises stresses obtained from the finite element models modelled using such structural parameters (H_{new}, h_{new}, etc). A correlation of adjusted $R^2 = 0.98$ was found. Figure 7 shows the estimated stress and calculated stress correlation plot. Figure 8 illustrates the sensitivity of each parameter in the stress distribution on the cortical bone around the implant.
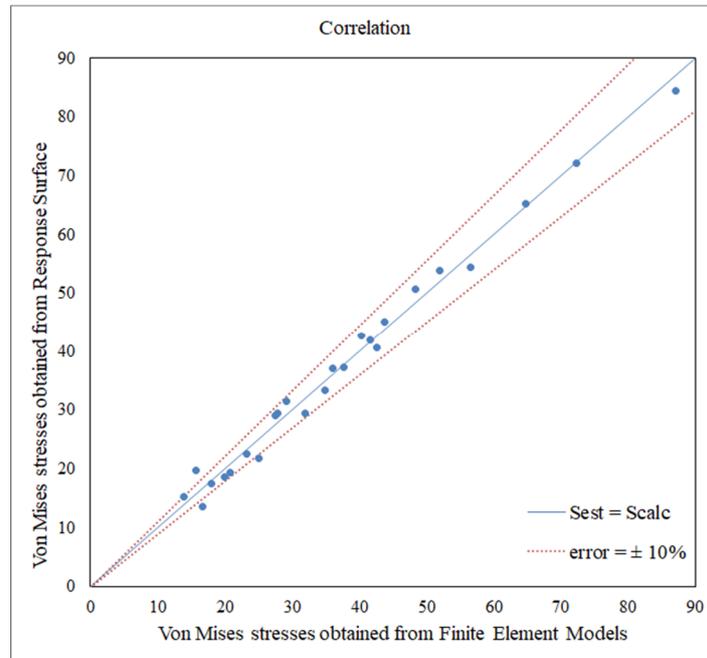

Figure 7. Estimated stress from response surface and calculated stress from finite element models correlation plot.
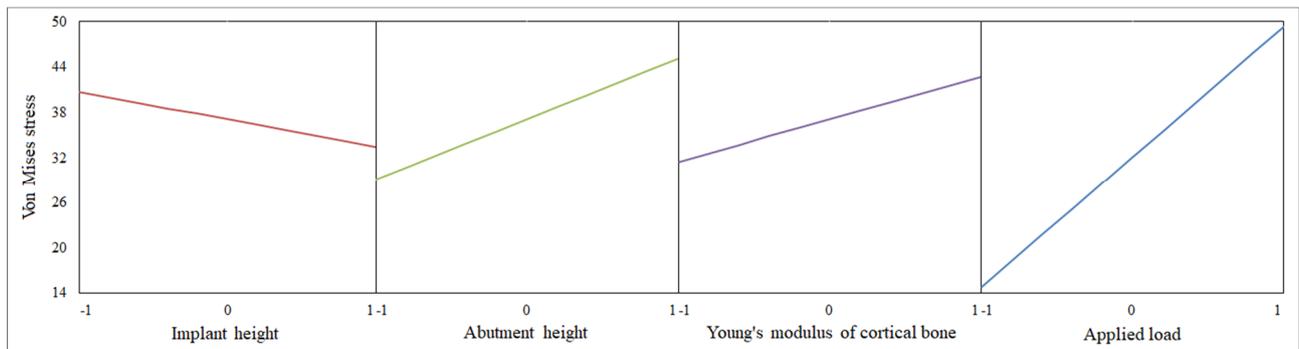

Figure 8. The sensitivity of each parameter in the stress distribution on the cortical bone around the implant.

4. CONCLUSION

A response surface representing the Von Mises stress distribution on cortical bone around the implant as a function of its structural parameters implant height (H), abutment height (h), Young's modulus of cortical bone (E), and applied load (F), was created. The response surface presented good correlation, all parameters behaved linearly and the most sensitive parameter is the applied load.

The use of Response Surface Methodology in the parametric analysis process sped up the analysis. An equation representing the von Mises stress on the cortical bone was obtained and predictions were made based on it, without the need of analysing every possible combination of parameters, which would result in new geometries, meshes and finite element calculations.

It is worth mentioning that, for simplification purposes, other conditions were not assessed in this study, such as the stress on the cancellous bone and the stress on the prosthesis. Nonetheless, this study adds confidence in the use of statistical techniques and finite element models to explore new designs of dental implants.

5. ACKNOWLEDGEMENTS

6. REFERENCES

Akça, K., Çehreli, M.C., and Iplikçioglu, H., 2002. "A Comparison of Three-Dimensional Finite Element Stress Analysis with In Vitro Strain Gauge Measurements on Dental Implants". *International Journal Of Prosthodontics*, Vol. 15, No. 2, pp. 115–121.

Albarracín, M.L., 2011. *Evaluation of the deformation of the abutment and perimplantar region as a function of the load applied to screw-retained implant-supported single crowns* (in Portuguese). Master's Dissertation, Graduate Program in Applied Dental Sciences, University of São Paulo - USP, Bauru, Brazil.

Aoki, T., Okafor, I.C.I., Watanabe, I., Hattori, M., Oda, Y., and Okabe, T., 2004. "Mechanical Properties of cast Ti-6Al-4V-XCu alloys". *Journal of Oral Rehabilitation*, Vol. 31, No. 11, pp. 1109–1114.

Baiamonte, T., Abbate, M.F., Pizzarello, F., Lozada, J., and James, R., 1996. "The experimental verification of the efficacy of finite element modeling to dental implant systems". *Journal of Oral Implantology*, Vol. 22, No. 2, pp. 104–110.

Craig, R.G., Powers, J.M., 1989. *Restorative dental materials*. Mosby, St. Louis.

Franciosa, P., and Martorelli, M., 2012. "Stress-based performance comparison of dental implants by finite element analysis". *International Journal on Interactive Design and Manufacturing*, Vol. 6, pp. 123–129.

Geng, J.P., Tan, K.B.C., Liu, G.R., 2001. "Application of finite element analysis in implant dentistry: A review of literature". *The Journal of Prosthetic Dentistry*, Vol. 85, No. 6, pp. 585–598.

Genna, F., 2003. "On the Effects of Cyclic Transversal Forces on Osseointegrated Dental Implants: Experimental and Finite Element Shakedown Analyses". *Computer Methods in Biomechanics and Biomedical Engineering*, Vol. 6, No. 2, pp. 141–152.

Hernandez, B.A., 2015. *Structural and fatigue analysis of implant-suported prosthesis by finite element method*. Master's Dissertation, Graduate Program in Mechanical Engineering, São Paulo State University – UNESP, Bauru, Brazil.

Juodzbalys, G., Kubilius, R., Eidukynas, V., and Raustia, A.M., 2005. "Stress distribution in bone: single-unit implant prostheses veneered with porcelain or a new composite material". *Implant Dentistry*, Vol. 14, No. 2, pp. 166–175.

Myers, R., Montgomery, D., and Anderson-Cook, C., 2009. *Response Surface Methodology, Process and Product Optimization Using Designed Experiments*. John Wiley & Sons, New Jersey.

Nist/Sismatech, 2012. e-*Handbook of Statistical Methods,* http://www.itl.nist.gov/div898/handbook. Accessed 18 November 2020.

Rangert, B., Jemt, T., and Jörneus, L., 1984. "Forces and moments on Branemark implants". *International Journal of Oral & Maxillofacial Implants*, Vol. 4, No. 3, pp. 241–247.

Skalak, R., 1983. "Biomechanical considerations in osseointegrated prostheses". *Journal of Prosthetic Dentistry*, Vol. 49, No. 6, pp. 843–848.

7. RESPONSIBILITY NOTICE

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