PARAMETRIZATION FOR RESPONSE SURFACE IN A DENTAL PROSTHESIS WITH FEM THREE-DIMENSIONAL MODELING

Gabriela Garcia Paschoeto, Mechanical Engineering Department – College of Engineering of Bauru – São Paulo State University (UNESP), Bioengineering Group, gabrielapaschoeto@outlook.com
Camilo Mesquita Neto, Mechanical Engineering Department – College of Engineering of Bauru – São Paulo State University (UNESP), Bioengineering Group, camilomn@gmail.com
Edson Antonio Capello Sousa, Mechanical Engineering Department – College of Engineering of Bauru – São Paulo State University (UNESP), Bioengineering Group, capello@feb.unesp.br

Abstract. *Dental implants have been widely used and have a great acceptance by patients, due to their capacity of performing the teeth function and esthetical issue. Since these natural system representation is complex or even unfeasible, model representation demand attention to the mechanical features and studies are still needed to avoid early failures. The purpose here was to develop parametric analysis of the structure of a three-dimensional implant, considering its material, geometry and loads, for a successful integration implant-bone. Response surface function was developed based on provided data (geometry, load and cortical bone stiffness) and as response Von Mises stress in cortical bone were obtained. More influential parameters were identified in the response. For this the concepts of Response Surface Methodology and Design of Experiments were applied with finite element models. The function obtained was 97,8% reliable and showed a faster estimation of critical stress, research of failures and accurate solutions.*

Keywords*: Biomechanics, Three-dimensional Dental Prosthesis, Finite Element Method, Parametric Analysis.*

1. INTRODUCTION

In recent years, engineering showed great enhancement of project techniques, innovative materials, new methods of solving fails, stables geometries, among others. In this situation, a new interdisciplinary knowledge area has merged, the Bioengineering. Knowledge of engineering area together with medicine, odontology and veterinary medicine developed new studies and equipment in order to help treatment and recovery of ill or injured patients. Currently, Bioengineering has shown great development and has potential to create innovative solution in the area.

Odontological prosthesis are used in substitution of natural teeth structure, which was extracted due to any reason. The mechanical behavior of these prosthesis is studied by Bioengineering area. The prosthesis function is to work as the natural teeth structure and be esthetically appropriate. Due to the great number of successful implants, their use has become really common among odontology specialists. Nevertheless, mechanical or biological problems are common and can cause some implant fails in the long term.

In order to improve odontological treatments and patient recovery, more studies and accurate modeling are needed. The Finite Elements method (FEM) can analyze structures with complex geometries, loads and features, and also non-linear situation of difficult experimenting (WAKABAYASHI et al., 2008). This method has been applied in odontological prosthesis development resulting in successful implants.

Basically, the MEF transforms a complex continuous implant structure into a discrete model of several elements. Based on the behavior analysis of each element, the behavior of the whole set can be understood. These elements are connected by discrete points, called nodes. The parameters that describe the behavior of the system are the nodal displacements. The use of MEF in Bioengineering is vast, besides nonlinear situations. In addition, computational models allow the solution of infinite problems with a much greater speed than analytical models.

Commonly new implants development is based on data of implants already available in the market. A parameter sensibility analysis can demonstrate which features are more influential in the model and could be modified to improve the traditional available prosthesis (FREITAS, 2016). Besides mechanical features, other properties can be changed for more possibilities of improvement, which are dimensions, shape, position, and others. The union of this analysis with the structural analysis method FEM allow the non-destructive study of many possible parameters set, to obtain an optimized implant model.

2. OBJECTIVES

This study aims to present a methodology for the parametric study of a unitary dental implant in relation to its biomechanical behavior of stress in the cortical bone from the variation of the implant geometric parameters (length, abutment length, cortical bone rigidity and intensity of the applied load). In this way, we propose the application of statistical methods to obtain a response surface that represents such behavior and also to describe the most influential parameters for the response. Thus, with the response surface equation, we can calculate the model's stress more quickly and practically.

3. METHODOLOGY

The geometry of the model developed in finite elements was based on the real dental prosthesis used by Albarracín (2011) and studied later by Hernandez (2015) and it is represented in Fig. (1).

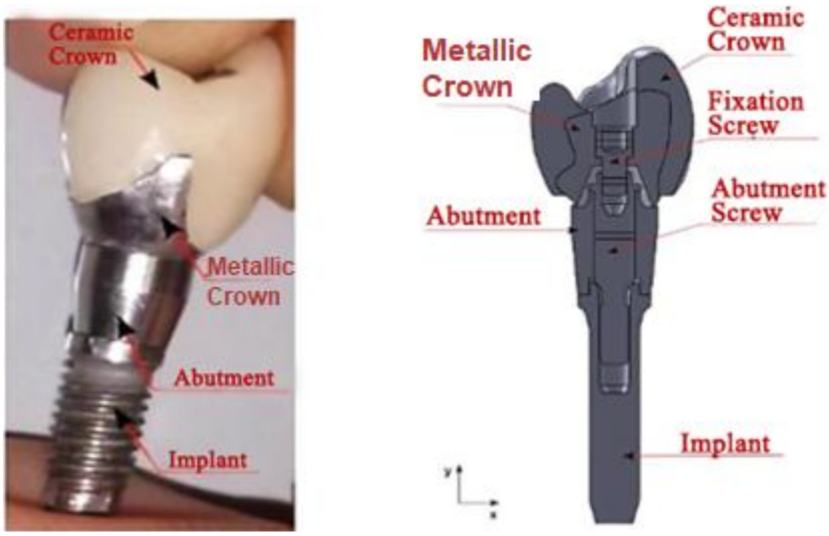

Figure 1. Real dental prosthesis (left) and computational dental prosthesis model in cut (right) (Hernandez, 2015)

For the development of this project a Branemark System Mk III Groov implant of 3.75 mm in diameter was obtained (Nobel Biocare - Göteborg, Sweden). A multi-unit abutment (Nobel Biocare - Göteborg, Sweden) was screwed onto the implant with a titanium screw. A prosthetic crown, cast in a cobalt-chromium alloy (Co-Cr) and covered by an external crown of feldspathic ceramics (CNG prosthetic solutions, São Paulo, SP, Brazil) was inserted with a prosthetic titanium screw.

The Finite Element program used in the preparation of three-dimensional mathematical models and for finite element analysis by Hernandez (2015) was Ansys (Ansys 15.0, Swanson Analysis System, Houston, Pa, USA). This program was also used at the work developed here for parameter changes and generation of new solutions.

The materials used in this study were considered homogeneous, isotropic and linearly elastic, as in Hernandez (2015), that is, they have the same composition and the same mechanical properties in all directions in a same point of the structural element and were characterized by modulus of elasticity and Poisson's coefficient. Table (1) shows the properties of the model components.

Table 1. Properties of the components used in the model

MATERIAL	MODULUS OF ELASTICITY E (GPA)	POISSON’S COEFFICIENT	REFERENCES
Cortical Bone	14	0,3	Juodzbalys *et al*. (2005)
Cancellous Bone	1	0,3	Juodzbalys *et al*. (2005)
Implant and abutment (Ti-6Al-4V)	110	0,34	Aoki *et al*. (2004)
Methalic Crown (Alloy of Co-Cr)	218	0,33	Craig (1989)
Fixation screw and abutment screw (Pure Ti)	100	0,34	Aoki *et al*. (2004)
Ceramic Crown (Feldspathic Ceramics)	68,9	0,28	Geng *et al*. (2001)

In the Finite Element mesh, the element used for the components discretization was SOLID187 of Ansys software. This element is a 3D solid, of quadratic interpolation function, it has ten nodes with three degrees of freedom per node. It was chosen because its function offers a greater agreement to the real conditions and has the characteristic of adapting to irregular meshes while maintaining its properties. Figure (2) shows this element and the complete discretized model.

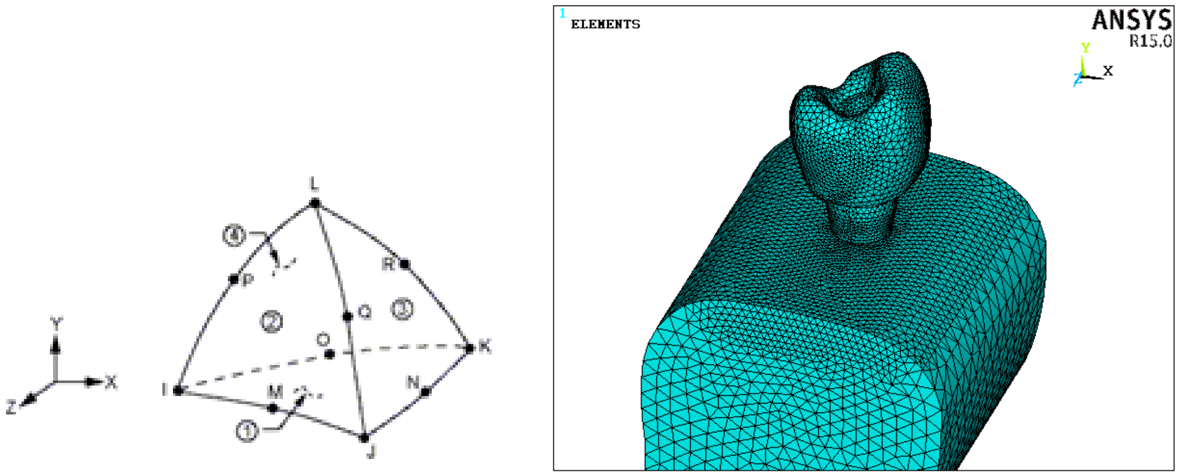

Figure 2. Element SOLID187 (left) and full discretized model (right)

The boundary conditions applied are shown in Fig. (3). A 30º tilted load was applied in the crown with intensity ranging from 30 N to 200 N for statistical analysis and movement restrictions were applied around the entire bone at all degrees of freedom.

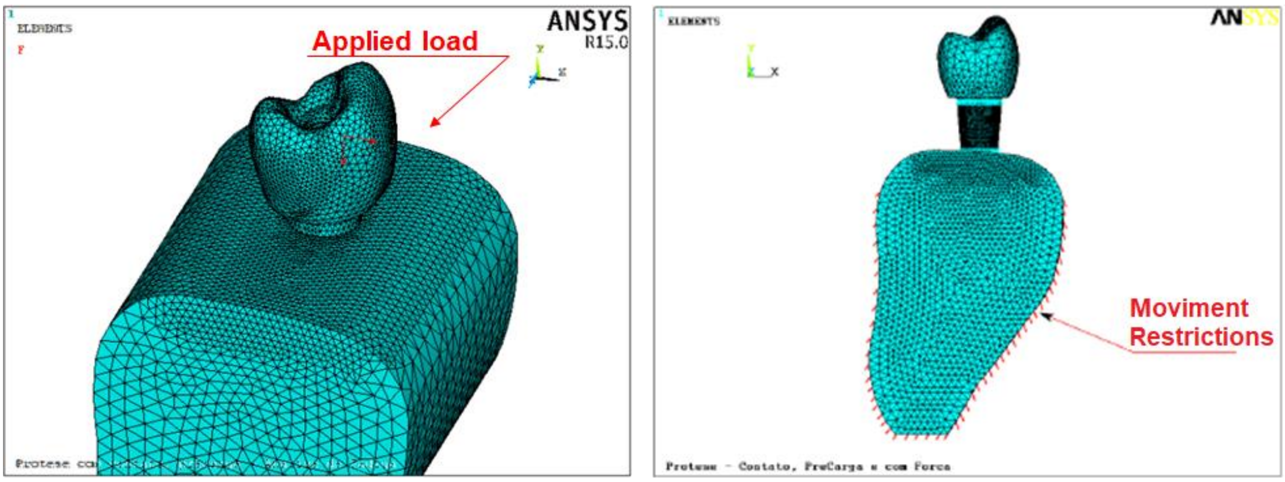

Figure 3. Load applied (left) and restrictions applied (right)

The parametric analysis applied in the set starts from a sensitivity analysis of the system, seeking a better understanding of what occurs mechanically at the bone interface with the implant in different situations defined by internal and external conditions to the biomechanical model. For this purpose, the Response Surface Methodology (RSM), Design of Experiments (DOE) (Handbook of Statistical Methods, 2012) and Microsoft Office Excel software were applied.

The DOE analysis chosen was the Central Composite Design. A central composite design is the most commonly used response surface designed experiment. Central composite designs are a factorial or fractional factorial design with center points, augmented with a group of axial points (also called star points) that let you estimate curvature. There are three types of Central Composite Design, the inscribed, the circumscribed and the face centered. This project uses the face centered type that has an alpha of 1. In this design the axial points are at the center of each face of the factorial space, so levels = + 1. This variety of design requires 3 levels of each factor (minimum, medium, maximum).

Figure (4) shows the parameters that were varied and Fig. (5) shows these parameters coded (implant length, X1; abutment length, X2; modulus of elasticity of cortical bone, X3; applied load, X4) and the sequence of analyses required for DOE (face centered composite design type).

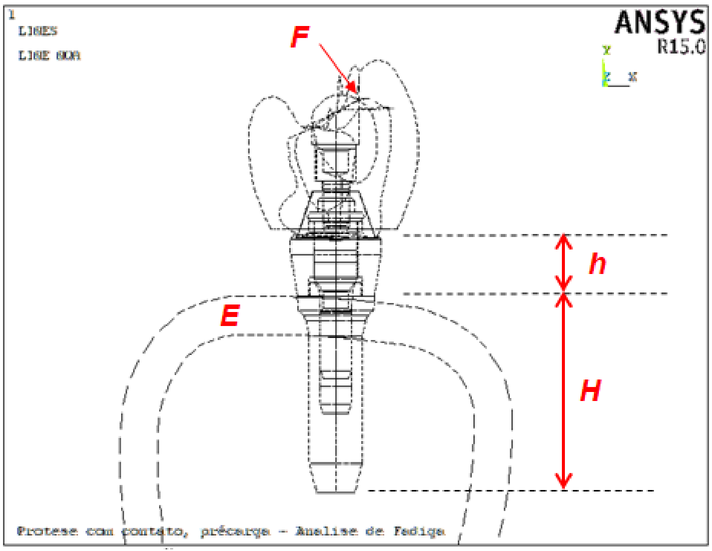

Figure 4. Parameterized parameters: implant length, H; abutment length, h; modulus of elasticity of cortical bone, E; applied load, F.

	Analysis
Variation of parameters	H, h, E, F (respectively)
Minimum	7 mm; 1 mm; $8x10^3$ MPa; 30 N
Maximum	15 mm; 5 mm; $20x10^3$ MPa; 200 N

Rodada	X1_cod	X2_cod	X3_cod	X4_cod	X1_real	X2_real	X3_real	X4_real
1	-1	-1	-1	-1	minimum	minimum	minimum	minimum
2	-1	-1	-1	1	minimum	minimum	minimum	maximum
3	-1	-1	1	-1	minimum	minimum	maximum	Minimum
4	-1	-1	1	1	minimum	minimum	maximum	maximum
5	-1	1	-1	-1	minimum	maximum	minimum	minimum
6	-1	1	-1	1	minimum	maximum	minimum	maximum
7	-1	1	1	-1	minimum	maximum	maximum	minimum
8	-1	1	1	1	minimum	maximum	maximum	maximum
9	1	-1	-1	-1	maximum	minimum	minimum	minimum
10	1	-1	-1	1	maximum	minimum	minimum	maximum
11	1	-1	1	-1	maximum	minimum	maximum	minimum
12	1	-1	1	1	maximum	minimum	maximum	maximum
13	1	1	-1	-1	maximum	maximum	minimum	minimum
14	1	1	-1	1	maximum	maximum	minimum	maximum
15	1	1	1	-1	maximum	maximum	maximum	minimum
16	1	1	1	1	maximum	maximum	maximum	maximum
17	-1	0	0	0	minimum	medium	medium	medium
18	1	0	0	0	maximum	medium	medium	medium
19	0	-1	0	0	medium	minimum	medium	medium
20	0	1	0	0	medium	maximum	medium	medium
21	0	0	-1	0	medium	medium	minimum	medium
22	0	0	1	0	medium	medium	maximum	medium
23	0	0	0	-1	medium	medium	medium	minimum
24	0	0	0	1	medium	medium	medium	maximum
25	0	0	0	0	medium	medium	medium	medium

Figure 5. Variation of the parameters (left) and standardized parameters and the sequence of analyses required for DOE (right)

By means of this parametric analysis a function called response surface was obtained and it describes the set stress in function of the selected parameters. It provides data on the geometry, load and stiffness, and show as response the Von Mises stress obtained in the cortical bone (Fig. (6)).

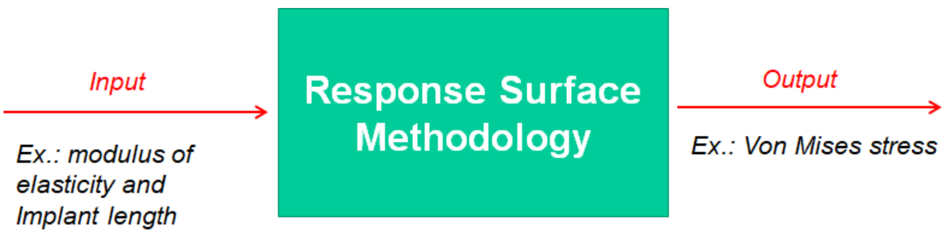

Figure 6. Representation of the response surface methodology

4. RESULTS AND DISCUSSIONS

After running the model with all parameter variations in Ansys software the Von Mises stress in the cortical bone (the implant region that suffers more wear) were obtained and it can be seen in Tab. (2).

Table 2. Von Mises Stress in the cortical bone obtained in the 25 analyses

Anlysis	S_real (Mpa)	Analysis	S_real (Mpa)	Analysis	S_real (Mpa)
1	8,9187	10	35,823	18	31,69
2	36,86	11	9,9337	19	17,468
3	14,378	12	50,027	20	45,132
4	49,666	13	8,7119	21	25,256
5	9,8075	14	62,329	22	35,939
6	71,006	15	12,621	23	4,9473
7	13,697	16	91,565	24	62,062
8	100,36	17	35,118	25	31,796
9	5,3246				

With the stress achieved, three distinct regressions were created on Microsoft Office Excel software to determine which response surface function better represented the model. These functions were a linear regression without interaction, a linear regression with interaction and a non-linear regression. The better results were obtained for the linear regression with interaction which is presented in Eq. (1) in function of the coded parameters (H, implant length; h, abutment length; E, modulus of elasticity of cortical bone; F, applied load).

The response surface obtained was presented and evaluated from the statistical point of view, presenting simultaneously the more and less influential parameters for the response. After the *p*-values analysis (that can be seen in Fig. (7)), the less significant terms can be removed (p-values larger then 0,05), which leads to a new response surface shown in Eq. (2).

$$S = -1{,}76589*H + 10{,}37947*h + 6{,}341667*E + 26{,}18657*F - 0{,}68307*H*h + 0{,}028088*H*E - 0{,}49613*H*F + 1{,}831888*h*E + 9{,}1626*h*F + 4{,}233313*E*F + 34{,}81747 \quad (1)$$

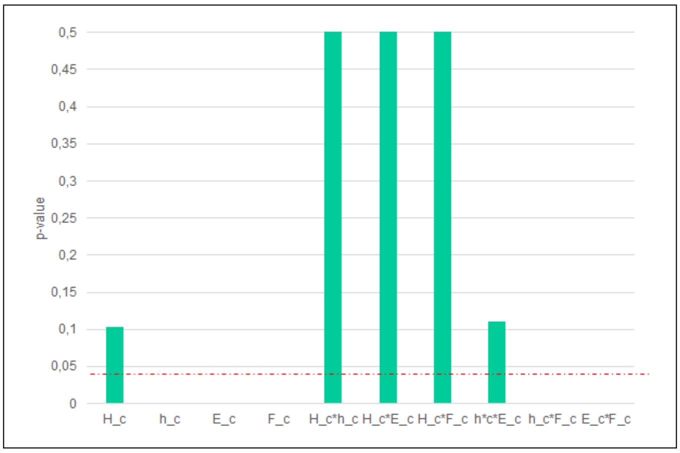

Figure 7. P-value of the normalized coefficients of the linear model with interaction

$$S = 10.38*h + 6.34*E + 26.19*F + 9.16*h*F + 4.23*E*F + 34.82 \quad (2)$$

The reliability level of Eq. (2) was 97,8%. Figure (8) shows a graphic comparing the values of stress obtained by the response surface function and the real stress values obtained by the FEM software, Ansys.

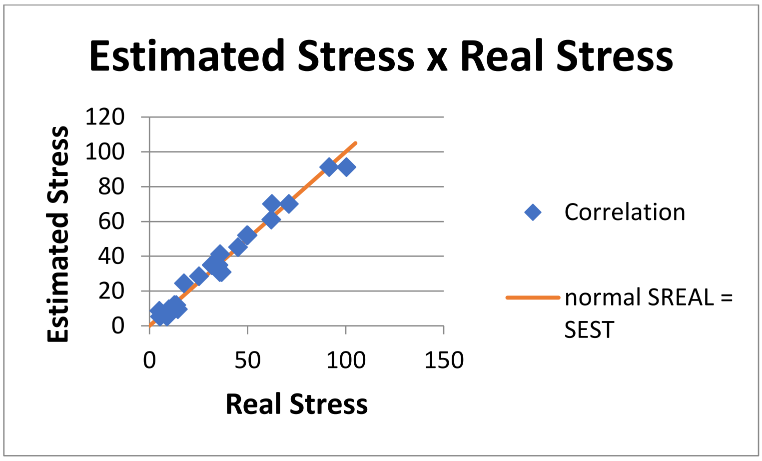

Figure 8. Estimated stress and real stress correlation graphic

5. CONCLUSIONS

The quality analysis of the generated response surface showed good representativeness when evaluated in relation to the statistical parameters of the model and the correlation with the real stress and the estimated stress by the function. Among all the analyzed parameters (implant length, abutment length, cortical bone stiffness and applied load), those that most affect the behavior of the cortical bone structure were the load and the length of the abutment.

From the obtained function, the influence of each parameter on the mechanical response in the bone were interpreted. Besides that, a more agile process in the estimation of critical stresses through functions were achieved. With the defined response surface, structural optimization solutions can be developed, aiming at minimizing stress in the cortical bone and at the same time reducing the area of bone removed, thus selecting better implants.

6. REFERENCES

ALBARRACÍN, M. L. **Evaluation of the deformation in the abutment and peri-implant region according to the load applied in screwed implant-supported single crowns.** 2011. 121 p. Master of Science Dissertation (Master of Science in Odontology) – Odontology College of Bauru, University of São Paulo, Bauru, 2011.

AOKI, T., OKAFOR, ICI., WATANABE, I., HATTORI, M., ODA, Y., OKABE, T. Mechanical Properties of cast Ti-6Al-4V-XCu alloys, **Journal of Oral Rehabilitation**, v.31 (11), p.1109-1114, 2004.

CRAIG, R. G.; POWERS, J. M. **Restorative dental materials**, 8. ed., St. Louis etc.: Mosby, 1989.

FREITAS, J. P. O. **Parametric Analysis and structural optimization of dental implant based on study of mechanical behavior of the bone-implant interface area**. 2016. 120 p. Master of Science Dissertation – Engineering College of Bauru, Sao Paulo State University, 2016.

GENG, J-P.; TAN, K. B.C.; LIU, G-R. Application of finite element analysis in implant dentistry: a review of the literature, **The Journal of Prosthetic Dentistry**, v. 85, n. 6, p. 585-598, 2001.

JUODZBALYS, G.; KUBILIUS, R.; EIDUKYNAS, V.; RAUSTIA, A. M. Stress distribution in bone: single-unit implant prostheses veneered with porcelain or a new composite material, **Implant dentistry**, v. 14, n. 2, p. 166-175, 2005.

HERNANDEZ, B. A. **Structural and Fatigue Analysis of Implanto-Suported Prosthesis by Finite Element Method.** 2015. 121 p. Master of Science Dissertation – Engineering College of Bauru, Post-Graduating Program, São Paulo State University, 2015.

NIST/SEMATECH. e-Handbook of Statistical Methods, 2012. Available in: <http://www.itl.nist.gov/div898/handbook/>. Access in: July 20. 2017.

WAKABAYASHI, N.; ONA, M.; SUZUKI, T.; IGARASHI, Y. Nonlinear finite element analyses: Advances and challenges in dental applications, **Journal of Dentistry,** v. 36, p. 463-471, 2008.

7. ACKNOWLEDGMENTS

We gratefully acknowledge financial support from CAPES and UNESP for all the support.

8. RESPONSABILITY FOR THE INFORMATIONS

The authors are solely responsible for the information included in this work.