23rdABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

EVALUATION OF PRE-LOAD VARIATION EFFECTS ON THE STRESS BEHAVIOR IN DENTAL PROSTHESIS: FINITE ELEMENT STUDY

Bruno Agostinho Hernandez
Alexander Paterno
Gabriela Garcia Paschoeto
João Paulo de Oliveira Freitas
Edson Antonio Capello Sousa
Engineering College of Bauru – São Paulo State University (UNESP), Bioengineering Group, Address: Eng. Luiz Edmundo C. Coube, Nº 14-01, Neighborhood: Vargem Limpa, Postal Code: 17033-360 – Bauru, São Paulo State, Brazil.
bhernandez@uol.com.br, alexander.paterno@hotmail.com, gabrielapaschoeto@outlook.com, joaopaulo.web@gmail.com, capello@feb.unesp.br.

Abstract.*The Finite Element Method (FEM) allows the analysis of problems with complex geometries and loads, hard experimentation, besides non-linear situations. Due to that, it has been used on the analysis of biomechanical systems, such as in dental prosthesis. In this last case, the higher stresses, which occur in the crown, implant and bone interfaces, the cyclic loads and the misfit between crown and implant, can lead to system failure. The misfit usually happens due to a lower torque (pre-load) applied in the moment of system setting. Therefore, this work aims to analyze the effects of torque variation on the stress distribution in the set, especially in the screws. So, in order to do the analysis, a Finite Element Model was developed. The model was composed by a single implant, abutment, abutment's screw, crown, fix's screw and mandible bone. Three oblique loads, with 30º from vertical, were applied: 100 N, 200 N and 280 N. The loads variation is due to the fact the masticatory forces may vary. For each load, three different torques (pre-loads) were used for the abutment's screw: 200 N.mm, 100 N.mm and 50 N.mm. The results demonstrate for the 200 N and 280 N loads, the stresses in the abutment's screw were higher than the yield stress for the screw. For the torque, it was noted the direct influence of it in the stress distributions.*

Keywords:*Finite Element Method, Dental prosthesis, Pre-load, Misfit.*

1. INTRODUCTION

The evolution of engineering in the last decades (new materials, new project techniques, new solution methods, etc) given rise a new research field, which is focused in the development of equipments that aim to help medicine in the treatment of people with serious diseases and injuries. This new field is called Biomechanical Engineering.

One of the studies which this new engineering does is about dental prosthesis. The dental prostheses are typical biomechanical structures because they have an objective to restore the mastication functions and they are responsible for replacing the original dental component that was damaged (Chen, *et al.*, 2011; Yang, *et al.*, 2007; Tabata, *et al.*, 2010). In other words, they are components that have the goal to restore the damaged mastication functions. These functions are a truly mechanical process because the dental structures are subjected to many different kinds of loads and answer to them. For this reason, the dental prostheses can be studied like structural components, which open the possibility to use the common procedures of analysis used in the engineering.

As a consequence of its mechanical characteristics, the dental prostheses are vulnerable to failure. One of the cases that may lead to failure is the incorrect adaptation or misfit.The misfit can occur due to inadequate torque that is applied on the screws. If the torque is lower, the system stability will be impaired and all the stresses equilibrium and distributions will be changed. This wrong stress distribution may lead to increase the stress in certain places which can start crack propagation. The cracks can provide an accumulation of bacteria, harming the well-being of the patient and the failure of whole system.So, a deep study about the characteristics and how the stresses are distributed in the system is fundamental to improve the safety and the quality of this kind of structure.

Although great advances were achieved in the past decades, many studies about dental prostheses involves situation where experimental procedures are hard to be conducted because dental structure works with complexes geometries, loads and mechanical behaviors, which an analytical solution is also difficult to achieve. Moreover, many problems in dental structures can´t be detected in any clinical or experimental procedures, like the misfit problem due to lower torque.

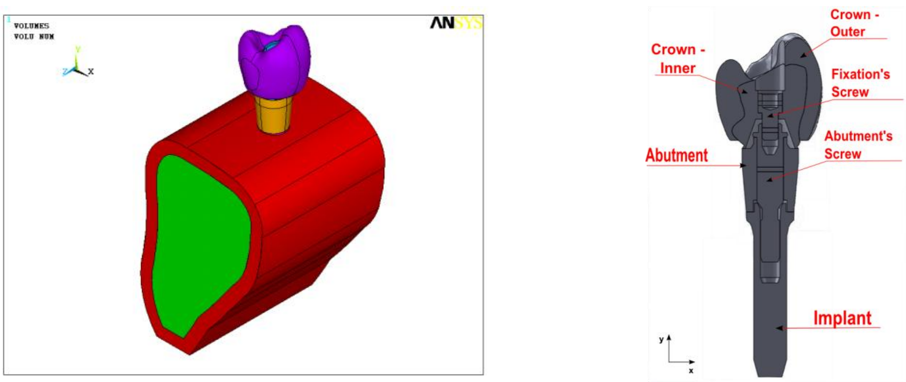
So, many researches in Biomechanical Engineering, especially in dental prostheses studies, are increasing the use of numerical methods, as the Finite Element Method – FEM (Chen, *et al.*, 2011; Clelland, *et al.*, 2009; Winter, *et al.*, 2009; El-Anwar, *et al.*, 2011; Perez, 2012; Tian, *et al.*, 2012).The FEM allows non-linear analysis and the study with complexes structures, materials, loads and behaviors. So, the use of FEM can help the understanding of failures in dental prostheses (Bae, *et al.*, 2012).

Furthermore, the objective of this study is to analyze the effects of torque variation on the stress distribution in the set, especially in the screws. In order to do the analysis, a Finite Element Model composed by a single implant, abutment, abutment's screw, crown, fix's screw and mandible bone was developed. Three oblique loads, with 30º from vertical, were applied: 100 N, 200 N and 280 N. For each load, three different torques (pre-loads) were used for the abutment's screw: 200 N.mm, 100 N.mm and 50 N.mm.

2. MATERIALS AND METHODS

2.1 CAD Modelling

For this study, a 3-D geometrical model of single implant-supported dental prosthesis Branemark System Mk III Groov with diameter of 3.75 mm and 13.0 mm length (Nobel Biocare - Göteborg, Sweden) was used. In this implant, an abutment, type multi-unit with 5.0 mm high (Nobel Biocare - Göteborg, Sweden) is screwed by a titanium screw. Above the abutment, a crown made of Co-Cr alloy and topped with feldspathic porcelain is also screwed by a titanium screw in the abutment (CNG Prosthetic Solutions, São Paulo, São Paulo State, Brazil). All this components were inserted in a mandible bone section. The crown and mandible bone section models were generated from Micro-CT scanning and CAD manipulation (Freitas,*et al.*, 2013 and 2014). The other components (abutment, screws and implant) were created just by CAD manipulation (Hernandez, 2015).The software CAD used was SolidWorks 2012. The Figure 1 lists the components display and the final model assembly.



(a) (b)

Figure 1. 3D Assembled model (a) and the list/display of the components (b).

2.2 Finite Element Analysis

2.2.1 Finite Element Modeling

The geometrical model was discretized in finite elements using the software Ansys (v.15.0 - Ansys Inc., Pennsylvania, USA, Module Mechanical APDL). The mesh was mapped to avoid bad elements. The element used to mesh was SOLID 187, from Ansys Library, which has 10 nodes, with 3 degree of freedom per node. There were generated 913318 elements and contact regions were inserted in the interfaces between the abutment and implant and between abutment and crown. The other interfaces were considered fully consolidated (bonded). The Figure 2shows the finite element model.

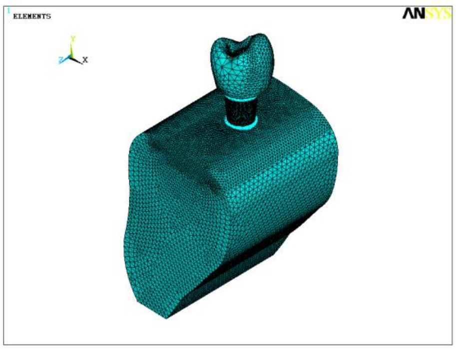


Figure 2. Finite Element model.

Each component was characterized by its material. Mechanical properties of materials used in this study are shown in Table 1. Behaviors of these materials are represented with linear isotropic and homogeneous material models.

Table 1. Mechanical properties of materials used in the study.

Material	*Model*	*Young Modulus (MPa)*	*Poisson*	*Ref.*
Pure Ti	Screws	100×10^3	0,34	Sakagushi, *et al.*, 1994
TI-6Al-4V	Implant Abutment	110×10^3	0,34	Sertgöz, *et al.*, 1996
Feldspathic Porcelain	Crown - outer	$68{,}9 \times 10^3$	0,28	Geng, *et al.*, 2001
Co-Cr	Crown - inner	218×10^3	0,33	Craig, *et al.*, 1989
Medullar Bone	Mandible	1×10^3	0,30	Juodzbalys, *et al.*, 2005
Cortical Bone	Mandible	14×10^3	0,30	Juodzbalys, *et al.*, 2005

2.2.2 Loading and Boundary Conditions

Due to mastication action, a dental implant will be under loads. These loads may vary between 91 N and 284 N (Petrie, *et al.*, 2007). So, in order to evaluate the load influence in the stress distribution, three different loads were selected: 100 N, 200 N and 280 N. The load was applied on the crownwith 30º, counterclockwise, in the point illustrated in Fig. 3. For the study of torque influence on the stress distribution in the set, especially in the screws, three different torques were selected for the abutment’s screw and tested for each load related before. The suggested torque by the manufacturersfor this screwis 200 N.mm. So, for this study, two other lowers torques were chosen: 50 N.mm and 100 N.mm. The torque for the fixation’s screw was the same for all the analysis: 100 N.mm. Finally, the boundary conditions for all the analysis also were the same: the down base and the lateral bone faces were restricted in three longitudinal directions and in three rotational directions. The Figure 4 illustrates the boundary conditions.

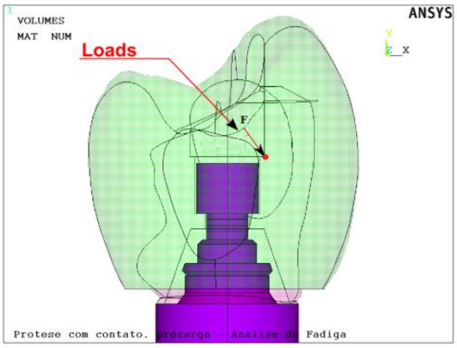


Figure 3. Applied loads.

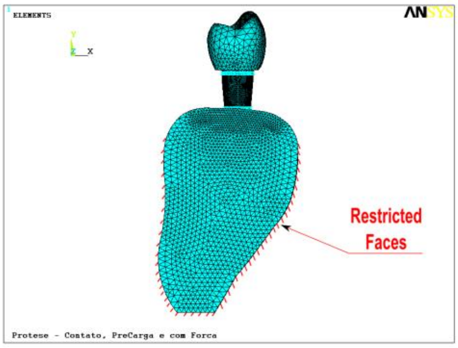


Figure 4. Boundaries conditions.

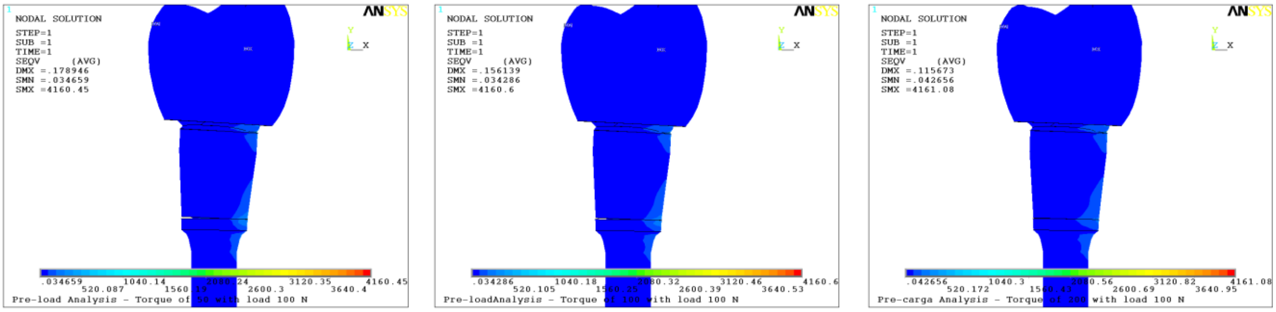
2.3 Points of Analysis

For this study, as related before, the abutment’s screwwas selected for the stresses analysis. It was selected due to the fact that it is responsible for the stabilization and the right stress distribution in the set (Tang, *et al.*, 2012).

Moreover, there are a high number of failures related to this part(Perez, 2012; Bae, *et al.*, 2012;Felli, *et al.*, 2011;Sannino, *et al.*, 2013; Hasan, *et al.*, 2014).The stress will be analyzed in a point upper the first thread.

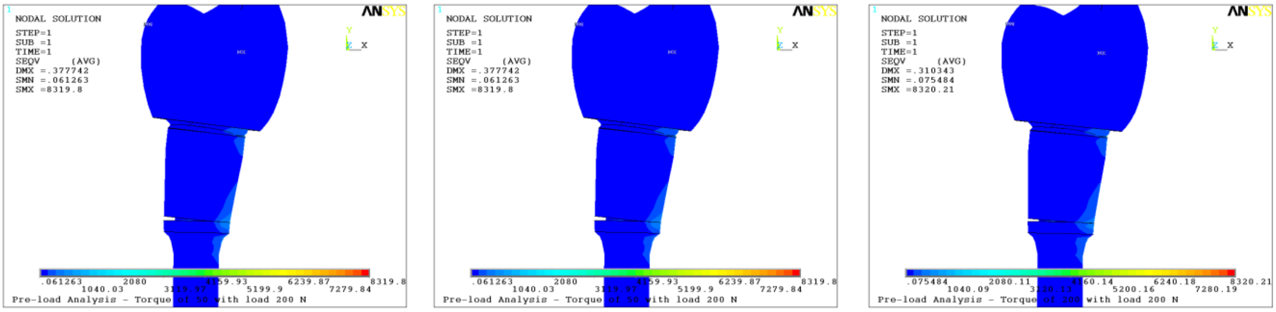
3. RESULTS

The Figures 5, 6 and 7 show the Von Mises Stresses and the misfit between the abutment and the implant for each load and torque types. The results are showed in 1:5 scales. The Table 2 lists the misfit for each load and torque typesand the Fig. 8 shows the relation between load, misfit and torque.



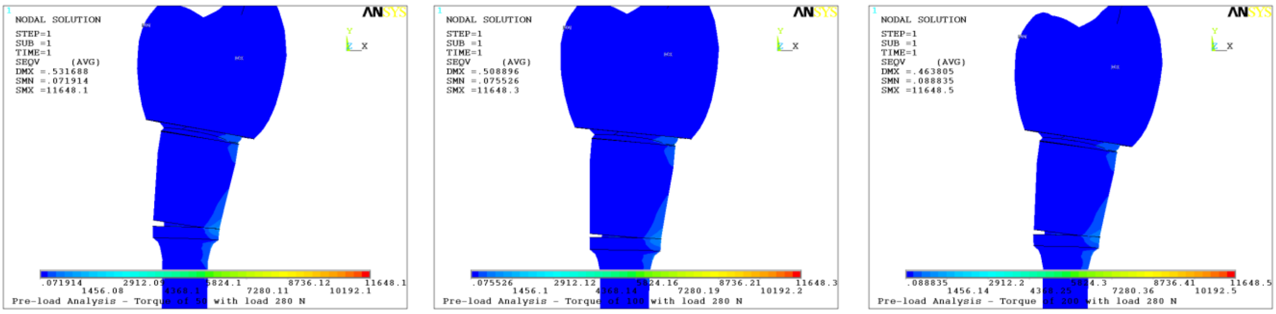
(a) (b) (c)

Figure 5. Von Misses Stresses [MPa] for the case of 100 N: (a) 50 N.mm, (b) 100 N.mm and (c) 200 N.mm.



(a) (b) (c)

Figure 6. Von Misses Stresses [MPa] for the case of 200 N: (a) 50 N.mm, (b) 100 N.mm and (c) 200 N.mm.



(a) (b) (c)

Figure 7. Von Misses Stresses [MPa] for the case of 280 N: (a) 50 N.mm, (b) 100 N.mm and (c) 200 N.mm.

Table 2. Misfit resultsfor each load and torque types in [mm].

	50 N.mm	*100 N.mm*	*200 N.mm*
100 N	0.0250	0.0195	0.0100
200 N	0.0560	0.0501	0.0392
280 N	0.0794	0.0738	0.0628

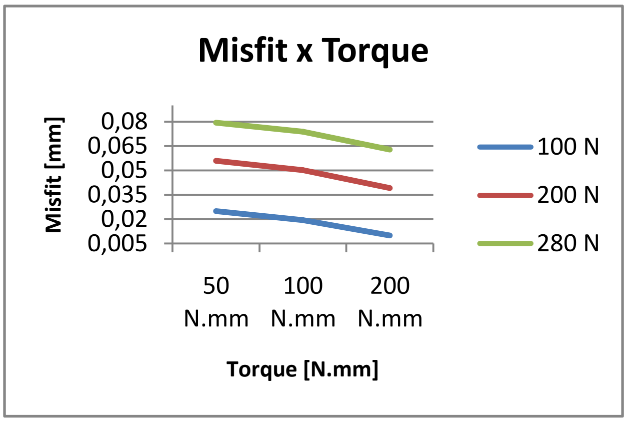
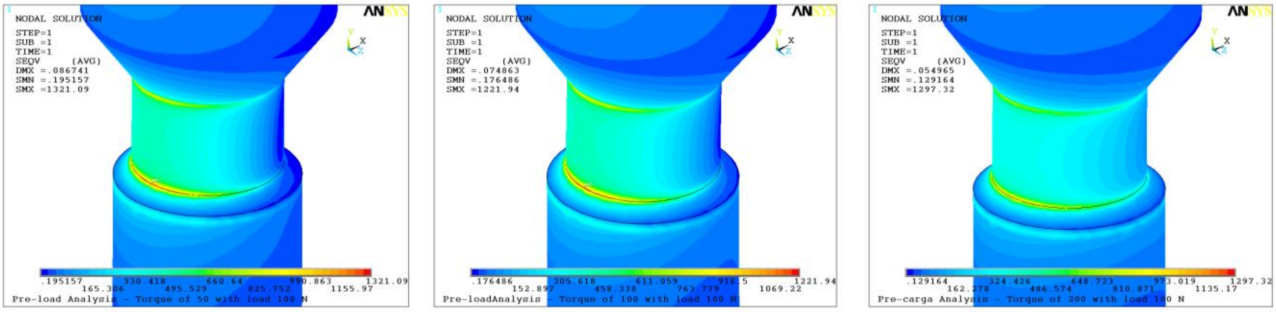


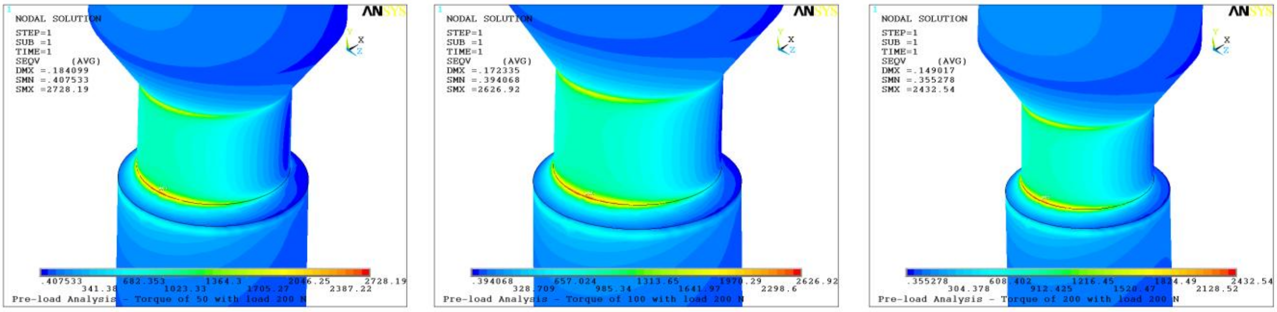
Figure 8. Relation between load, misfit and torque

The Figures 9, 10 and 11 show the Von Mises Stresses for the abutment's screw. The Table 3 lists the Von Mises Stresses for each load and torque types in a point upper the first thread and the Fig. 12 shows the relation between load, stress and torque.



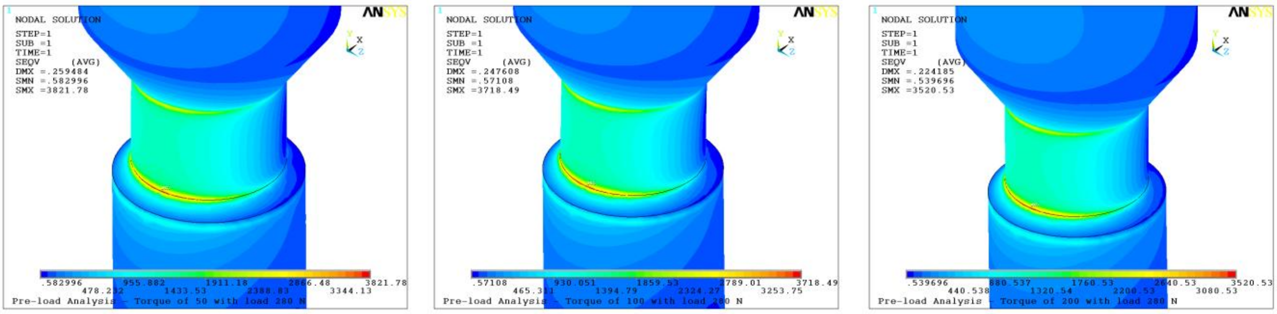
(a) (b) (c)

Figure 9. Von Misses Stresses [MPa] for the case of 100 N: (a) 50 N.mm, (b) 100 N.mm and (c) 200 N.mm.



(a) (b) (c)

Figure 10. Von Misses Stresses [MPa] for the case of 200 N: (a) 50 N.mm, (b) 100 N.mm and (c) 200 N.mm.



(a) (b) (c)

Figure 11. Von Misses Stresses [MPa] for the case of 280 N: (a) 50 N.mm, (b) 100 N.mm and (c) 200 N.mm.

Table 3. Von Misses Stresses for each load and torque types in a point upper the first thread in MPa.

	50 N.mm	*100 N.mm*	*200 N.mm*
100 N	585.9	538.1	471.3
200 N	1213.7	1164.8	1071.1
280 N	1700.7	1651.1	1558.8

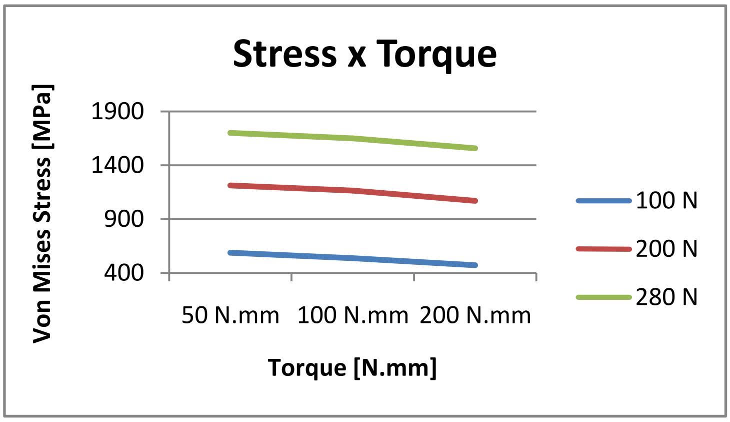


Figure 12. Relation between load, stress and torque.

4. DISCUSSION

The evolution of computers technology and engineering techniques allows the development of new studies and new researches areas. New approaches could be suggested and evaluated. Moreover, new and old project methods could be used easily and faster like the Finite Element Method (FEM). The automation of this specific method allowed the simulation and the analysis of a large quantity of structures and more complexes solutions could be proposed.

One example about the efficiency of this method in the evaluation of difficult situations is showed in this study. Using finite element analysis, the misfit between the abutment and the crown could be measured and evaluated. In a simple clinical or experimental analysis it couldn’t be measured.Analyzing the Figs. from 5 to 7, it can be noticed that when the torque value decreases the misfit between abutment and implant increase. The condition with the smallest torque (50 N.mm) presented the higher misfit, with 0.0794 mm, 0.0560 mm and 0.025mm of misfit for the cases with 280 N, 200 N and 100 N, respectively. The best conditions were presented for the 200 N.mm torque, which is recommended by the manufactures, with 0.010 mm, 0.0392 mm and 0.0628 mm of misfit for the cases with 100 N, 200 N and 280 N, respectively.

So, according with the results, the best case is to use the recommended torque, 200 N.mm, once it presented the lower misfit between the cases. The importance in the control of correct torque is because as greater the misfit is the probability of bacteria accumulation in that place increase. This accumulation can lead to a tissue inflammation and to problems in the osseointegration, harming the patient’s health. Furthermore, a high misfit value can change the stress distribution in the set and overload some components.

It also can be observed from Figs. 5 to 7 that there is a contact stress between the abutment and the implant and between the abutment and the crown for all the cases. This contact stress can be harmful because it can induce cracks in the implant’s surface which can give rise a total failure of the system. Nevertheless, the contact stress also illustrates the functionality of the model and the contact elements in the simulation of biomechanical conditions.

Analyzing the results for the screw from Figs.9 to 11, higher stress levels were noted in first thread region for all the cases. This area is commonly cited in the literature because of the higher number of failures related (Felli, *et al.*, 2011; Bae, *et al.*, 2012; Pérez, 2012; Sannino, *et al.*, 2013; Hasan, *et al.*, 2014). The worst case was for 280 N load and 50 N.mm torque, with 1700 MPa. The best situation was for the case with 100 N load and 200 N.mm torque, with 471.3 MPa. This is in an agreement with the literature because the 100 N is anusual loadand it is used in many studies (Bilhan, *et al.*, 2014). However, among the torque cases for the 100 N load, just for the case with 200 N.mm the stress value was fewer than the Yield Modulus of Titanium, which is 480 MPa (Lütjering, *et al.*, 2003). For the other cases, all the stresses values were higher than that value. These results show the importance in the applying of the correct torque in the screws. A wrong torque can lead a higher misfit and an overload which may cause higher stresses in the screws, occurring, sometimes, a plastic deformation in the screw.

Analyzing all the set and the stresses distributions, it is noted that the abutment’s screw is the most important structure in the set because it is responsible to fix the abutment and the crown in the implant. So, any problem with the abutment’s screw can lead to failure in the system.

The Figures 8 and 12 and the Tables 2 and 3 summarize all the information acquired. It can be seen a direct relation between stress and torque and between the misfit and torque. The increase in the torque decreases the misfit and the stresses levels. On the other hand the increase in the load will increase the misfit and the stresseslevels.

5. CONCLUSION

In this study, a finite element model of an implant-supported dental prosthesis was developed in order to study the influence of torque and load in the stresses distributions in the system, mainly in the abutment's screw. This study has observed that the increase in the torque decreases the misfit and the stresses levels and the increase in the load will increase the misfit and the stresses levels. Moreover, it was shown that the best situation for the implant system was with 200 N.mm torque (which is recommended by the manufactures) and with 100 N load (commonly used load in many studies). Finally, this study demonstrated the capacity of Finite Element Method has to reproduce the real conditions of biomechanical behavior.

6. ACKNOWLEDGEMENTS

Acknowledgments to CAPES for Master of Science Scholarship.

7. REFERENCES

Ansys. *User's manual for revision 6*. Swanson Analysis Systems Inc., Houston, PA. 2001.

Bae, J-S.; Jeong, H-Y.; Effects of material properties and hole designs of the jig on the fatigue life of dental implants. *Journal of Mechanical Science and Technology*, v.26, p. 759-766, 2012.

Bilhan, S. A., Baykasoglu, C., Bilhan, H., Kutay, O., &Mugan, A. Effect of attachment types and number of implants supporting mandibular overdentures on stress distribution: A computed tomography-based 3D finite element analysis. *Journal of Biomechanics*, 2014.

Chen, L-J.; He, H.; Li, Y.; Finite element analysis of stress at implant–bone interface of dental implants with different structures, *Transactions of Nonferrous Metals Society of China*,v. 21, p. 1602-1610, 2011.

Clelland, N. L., Seidt, J. D., Daroz, L. G., & Mcglumphy, E. A. Comparison of strains for splinted and nonsplinted implant prostheses using three-dimensional image correlation. *The International Journal of Oral & Maxillofacial Implants*, v. 25, n. 5, p. 953-959, 2009.

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

El-Anwar, M.I.; El-Zawahry, M.M. A three dimensional finite element study on dental implant design, *Journal of Genetic Engineering and Biotechnology*, v. 9, p. 77-82, 2011.

Felli, F.; Pilone, D.; Scicutelli, A.; Fatigue behavior of titanium dental endosseous implants. *Fratturaed Integrità Strutturale*, v.18, p. 14-22, 2011.

Freitas, J. P. O. ; Hernandez, B. A., Albarracín, M. L. Sousa, E. A. C. Reverse engineering on acquisition, processing and reconstruction of images for three dimensional remodeling applied to calculus with finite element (ISSN 2014-0066). In: VIII Congresso Nacional de Engenharia Mecânica, 2014, Uberlândia – Minas Gerais. CONEM, 2014.

Freitas, J. P. O. ; Sousa, E. A. C. ; Foschini, C. R. ; Santos, R. R. ; Rahal, S. C. Pretreatment and reconstruction of three-dimensional images applied in a locking reconstruction plate for structural analysis with FEA. In: *Computational Vision and Medical Image Processing IV - VIPIMAGE 2013, 2013*, Madeira Island. Computational Vision and Medical Image Processing, 2013. v. 4.

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.

Hasan, I., Keilig, L., Bourauel, C., & Lückerath, W. The Effect of Screw Preload and Framework Material on the Success of Cementable Fixed Partial Prostheses: A Finite Element Study. *Annals of Anatomy-Anatomischer Anzeiger*, 2014.

Hernandez, B.A.; Capello Sousa, E. A. Analysis of the contact mechanic problem by FEM using bi and three dimensional models. Partial and Final Scientific Initiation Report to CAPES Scholarship(PIBIC –UNESP), Bauru - SP, 2010.

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.

Lütjering, G.; Williams, J. C. Titanium: engineering materials and processes. *Springer*, Manchester, UK, p. 1-356, 2003.

Pérez, M. A. Life Prediction of Different Commercial Dental Implants as Influence by Uncertainties in their Fatigue Material Properties and Loading Conditions. *Computer Methods and Programs in Biomedicine*, v. 108, p.1277-1286, 2012.

Petrie, C. S.; Williams, J. L. Probabilistic analysis of peri-implant strain predictions as influenced by uncertainties in bone properties and occlusal forces. *Clinical oral implants research*, v. 18, n. 5, p. 611-619, 2007.

Sakaguchi, R. L.; Borgersen, S. E. Nonlinear contact analysis of preload in dental implant screws. *The International Journal of Oral & Maxillofacial Implants*, v. 10, n. 3, p. 295-302, 1994.

Sannino, G.; Barlattani, A.; Mechanical Evaluation of an Implant-Abutment Self-Locking Taper Connection: Finite Element Analysis and Experimental Tests. *The International Journal of Oral & Maxillofacial Implants*, v.28, p. 17-26, 2013.

Sertgöz, A.; Güvener, S. Finite element analysis of the effect of cantilever and implant length on stress distribution in an implant-supported fixed prosthesis. *The Journal of Prosthetic Dentistry*, v. 76, n. 2, p. 165-169, 1996.

Tabata, L. F., Assunção, W. G., Barão, V. A. R., Gomes, E. A., Delben, J. A., De Sousa, E. A. C., & Rocha, E. P. Comparison of single-standing or connected implants on stress distribution in bone of mandibular overdentures: a two-dimensional finite element analysis. *Journal of Craniofacial Surgery*, v. 21, n. 3, p. 696-702, 2010.

Tang, C-B.; Liu, S-Y.; Zhou, G-X.; Yu, J-H.; Zhang, G-D; Bao, Y-D.; Wang, Q-J; Nonlinear finite element analysis of three implant – abutment interface designs, International Journal of Oral Science, v. 4, p. 101-108, 2012.

Tian, K.; Chen, T.; Han, L.; Yang, J.; Huang, W.; Wu, D.; Angled abutments result in increased or decreased stress on surrounding bone of single-unit dental implants: A finite element analysis, *Medical Engineering & Physics*, v. 34, p. 1526-1531, 2012.

Winter, W., Mohrle, S., Holst, S., & Karl, M. Bone loading caused by different types of misfits of implant-supported fixed dental prostheses: a three-dimensional finite element analysis based on experimental results. *The International Journal of Oral & Maxillofacial Implants*, v. 25, n. 5, p. 947-952, 2009.

Yang, J.; Xiang, H-J. A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone. *Journal of Biomechanics*, v. 40, n. 11, p. 2377-2385, 2007.

8. RESPONSIBILITY NOTICE

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