https://www.scirp.org/journal/jbise

J. Biomedical Science and Engineering, 2021, Vol. 14, (No. 6), pp: 223-232

Analysis of the Influence of Parafunctional Loads on the Bone-Prosthesis System: A Non-Linear Finite Element Analysis

Henry Figueredo Losada[1], Edison Gonçalves[2], Jose Luis Valin[3], Luis Ide[4]

[1]Faculty of Engineering, University of the Republic, Montevideo, Uruguay; [2]Polytechnic School, University of Sao Paulo, Sao Paulo, Brazil; [3]School of Mechanical Engineering, Pontifical Catholic University of Valparaíso, Valparaíso, Chile; [4]Faculty of Dentistry, University of Sao Paulo, Sao Paulo, Brazil

Correspondence to: Henry Figueredo Losada, henryf@fing.edu.uy

Keywords: Biomechanics, Analysis Non-Linear, Dental Implant, Finite Element Method

Received: April 9, 2021 **Accepted:** June 5, 2021 **Published:** June 8, 2021

ABSTRACT

The present study evaluates the effects of occlusal loading on an implant-supported dental implant with external hexagon dental implant-abutment systems, using the finite element method analysis. Tensile analyses were performed to simulate different axial and obliquous masticatory loads. The influence of the variations in the contouring conditions of the interfaces was analyzed to weigh the osseointegration with linear and non-linear cases, by means of a parametric design. The geometry selected to place the prostheses was a jaw section, considering the properties of the set of cortical and trabecular bones. The results show that for non-linear contour conditions, the stress presents smaller value distributions and signals a different place in the screw-implant interface as the factor of the greater weight in this study. The location indicated that von Mises stress concentrations are not exclusive to the contact regions studied, moving to an area that is not in direct contact with the non-linear contact interfaces. In addition, the direction of load with an angle of 15 degrees presented the highest values of von Mises stress.

1. INTRODUCTION

The most common complications in the use of implant-supported prostheses on all single-tooth restorations are related to the biomechanical conditions of the interfaces.

Data according to several authors suggests the most common causes of failure for implant-supported prostheses are loss of tightness in the upper abutment screw, which precedes the fracture of the implant, at around 7% [1]. For external hexagonal systems [2] finds failure rate to be approximately 38%. About 5.3%

of total implants fail within first year, while screw loosening rate is around 5.8% - 12.7% after five years [3]. It was reported that around 26% - 43% of intermediate screw required retightening in the first 5 years and 11% of screws loosened in 10 cases [2, 4]. Several studies show mechanical complications with external hexagonal connections, as they present a limited resistance to oblique parafunctional loads. In the application of preload in non-passive bolted prostheses, they generate bending moments and axial forces in osseointegrated systems, producing overloadings and a fracture of components and intermediate screws, and also microfractures of the trabecular bone resulting in the functional loss of the implant [5]. Previous studies using the finite element method show differences in values and distribution of tensions between the types of interfaces in the contact for fixed bond, slip contact, contact friction and non-linear contact. These simulations of finite elements with non-linear contact conditions present an advantage for evaluating the vital parameters in the components of the prosthesis [6, 7]. A friction coefficient in the attrition interface ($\nu = 0.3$) increases the tensions between 28% - 63% if compared to a bonded interface.

The aim of this study was to evaluate the influence of a load mastication under conditions of linear and non-linear contact interface, and to assess the distribution of tensions in the assembly of a prosthesis implant supported with external hexagon dental implant-abutment systems, through the finite element method.

2. MATERIALS AND METHODS

A CAD (3D) model of an implant-supported dental implant with external hexagon and bolted connection was generated by Inventor 2017 software (Autodesk) by acquiring data from a 3D scanner to generate surfaces (NURB) and reverse engineering of parts with a microscope. The model of the implant was integrated into the section of the mandible obtained by a CT [8], with dimensions of 10 mm in length, 25 mm of frontal height and 21 mm in final height. Dimensions obtained for approximate cortical bone were 1.88 mm to 2 mm in the trabecular bone with dimensions 4 mm up to 6.50 mm for a total width of approximately 11 mm in the pre-molar region. The geometries of the cortical and trabecular bone regions were considered. Implants were placed exactly at the same distance relative to the most coronal section of the cortical bone with surgical recommendations [9].

2.1. Finite Element Modeling

The assembled model was imported into ANSYS v16 software (Canonsburg, PA, USA) considering three regions of the contact for the analyses. **Figure 1** a shows surfaces chosen to insert the contact elements. The contact elements selected for the surfaces were CONTA175 and TARGET170 [10], with the ability to simulate plasticity and non-linearity. The FEM model is formed by the element SOLID87, which is a 10-node tetrahedron with three degrees of freedom from the Ansys library (ANSYS Inc., Canonsburg, USA), the interpolation function is quadratic; the geometric shape of the element adapts the irregular geometries and ensures the geometric characteristics. A numerical convergence analysis was performed to determine the sensitivity of the model to different mesh densities in the stress distributions. The numerical convergence criteria were used to obtain a value less than 5% in the difference in the value of the von Mises stress.

The total number of tetrahedron elements was 2,456,157, with a total of 90,969 contact elements in the regions shown in **Figure 1**. A summary of the components in contact are shown in **Table 1**.

For the representation of the tightening forces in the modeling, a residual stress was introduced into the screw, generating a reaction that traction the body. [1, 11, 12]. Calculating was done on the clamping force on the intermediate screw by the Equation (1), its diameter (d) was 2.00 mm and the torque usually employed in this screw was 200 N-mm [13]. The value to generate the tightening force (traction) was 686 N. The behavior of the materials was represented by a linear and isotropic behavior model. The mechanical properties of the materials used are shown in **Table 2**.

$$F_i = \frac{T}{0.20 \cdot d} \tag{1}$$

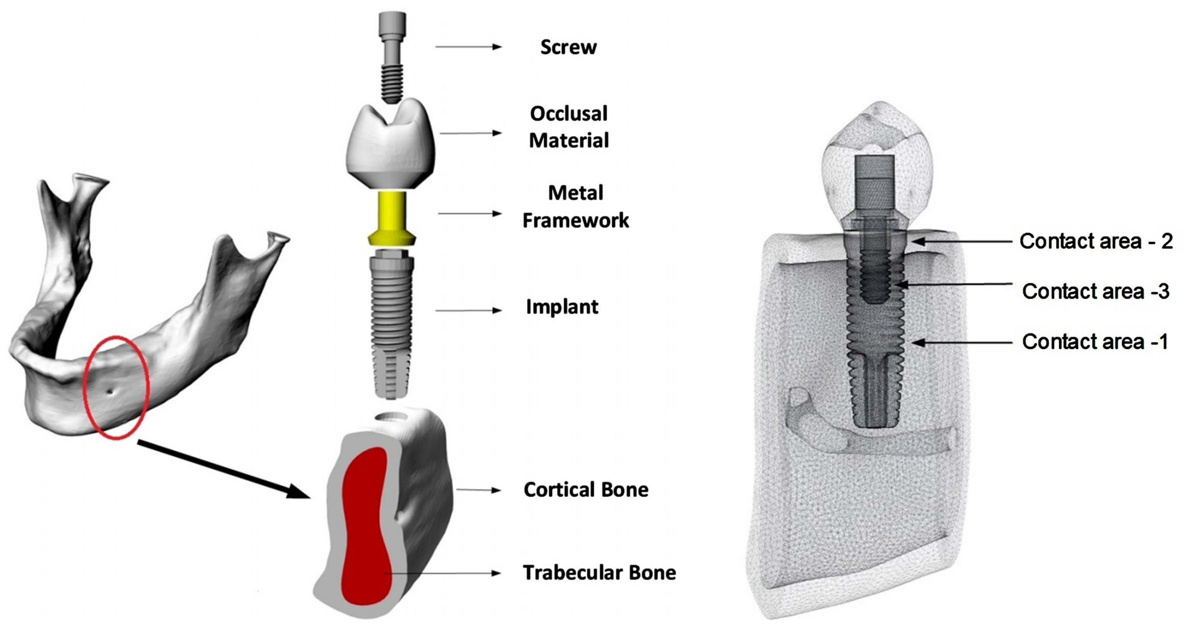

Figure 1. Selection and dimensions of the mandibular section for fitting the prosthesis—relation of the components of the prothesis. The mandible was reproduced by a TAC [8]; Regions of interest contact bone-implant interface.

Table 1. Summary of the contacts between all components.

Connected pairs Type of the contact		
Implant	Trabecular bone	Frictional
Implant	Cortical bone	Frictional
Screw	Implant	Frictional
Screw	Metal framework	Perfectly bonded
Trabecular bone	Cortical bone	Perfectly bonded
Oclussal material	Metal framework	Perfectly bonded

Table 2. Mechanical properties of materials used in the study.

Material	Properties		
	Young's modulus (MPa)	Poisson ratio	Reference
Ti-6Al-4V/Screw	110,000	0.32	[14]
Porcelain	68,900	0.19	[15]
Cortical bone	13,400	0.31	[16]
Trabecular bone	1370	0.31	[17]
Alloy (Co-Cr)	220,000	0.30	[12]

2.2. Loading Conditions

Static analysis of prostheses is necessary to ensure project safety. A total of nine cases of loading between axial and oblique loading simulating mastication were performed. Crown occlusion surfaces were set at 15-degrees intervals in the directions (see **Table 3**). A force measured clinically with a value equal to 100 N was applied to a rigid element defined by a sphere of 4 mm in diameter. This application of the load was defined by the contact between the sphere-crown of the prosthesis (see **Figure 2** and **Table 3**). In the section, the mandible at the base was restricted in all its directions.

3. NUMERICAL RESULT

The von Mises stress distributions along the implant-trabecular bone, implant-cortical bone and implant-screw interfaces were studied for an angle value $\alpha = 90^\circ$ (**Table 3**) in a state without friction and friction ($\mu = 0.3$), considering the stress values and location of the peak values (**Figure 3**). That is, the point marked "Max" was determined to be the highest value or critical point for the two states. The maximum stress value was 864 MPa in the frictionless state was observed at a point between the implant-screw interface on the first edge of the first screw thread. For the second case, considering the friction between the interfaces, the maximum stress value was 773 MPa. The location indicated that von Mises stress concentrations are not exclusive to the contact regions studied, moving to an area that is not in direct contact with the non-linear contact interfaces.

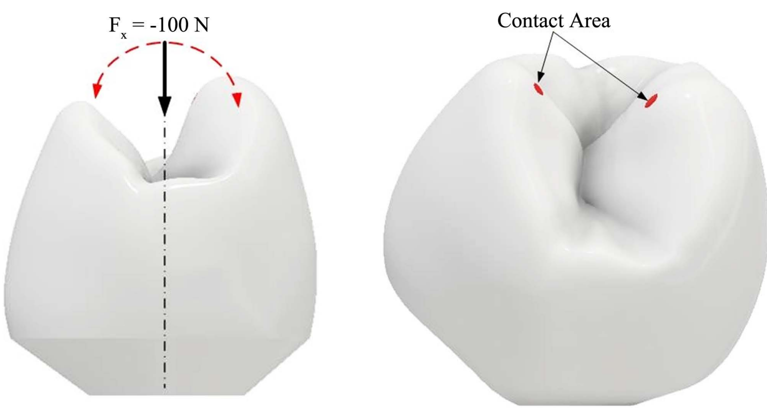

Figure 2. Representation of axial and oblique loading.

Table 3. Values of load angles evaluated.

Angle value (α)	Force X (N)	Force Y (N)
90˚	−100	0
60˚	−50	−86.60
45˚	−70.71	−70.71
30˚	−86.60	−50
15˚	−25.88	−96.59
−60˚	−50	86.60
−45˚	−70.71	70.71
−30˚	−86.60	50
−15˚	−96.50	25.88

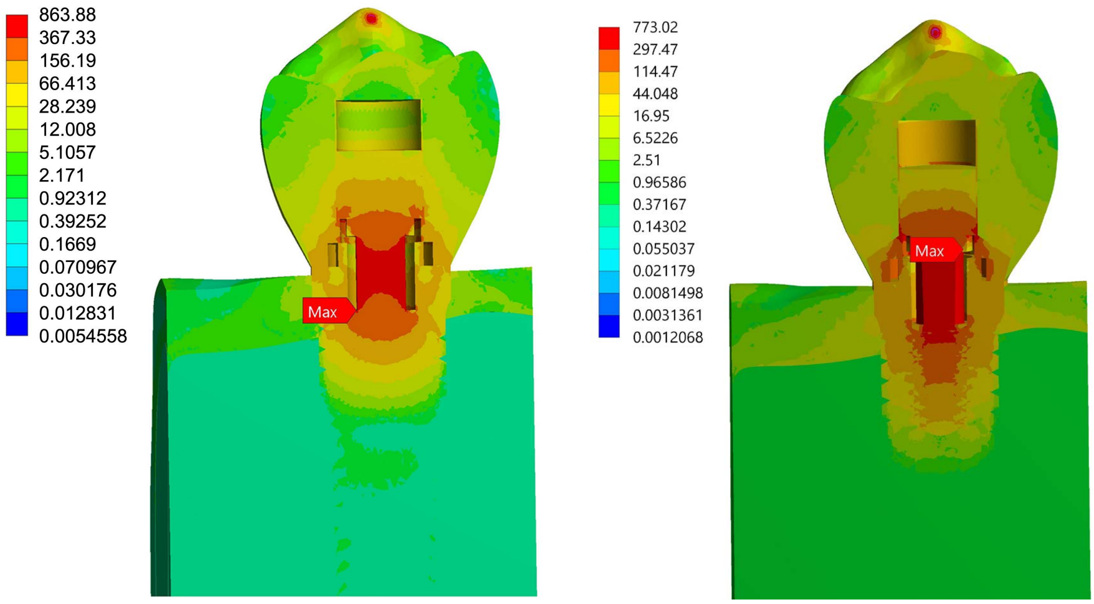

Figure 3. Distribution of von Mises stresses for bonded and friction conditions on an axial load, respectively.

The stress distributions of von Mises in a sagittal cut for axial loading are shown in **Figure 3**. The maximum stress values are above, the propagation of the stress occurs up to the first or second thread of the implant.

The stress distributions of Max Principal, Min principal and von Mises are evaluated at the highest critical point considering the different angular values (**Table 3**). Comparing the result of the von Mises stress analysis on the implant vs all the angular values (**Table 3**), the maximum stress value found was 550 MPa for loading with angle of 15˚, about 12% higher than referent angle value 90˚ (**Figure 4**).

Figure 5 represents the distribution of the stress in the Screw for all values of load angles. Maximum value was 567 MPa for loading with angle of 15˚, about 16% higher than reference load angle ($\alpha = 90˚$).

Finally, **Figure 6** represents the distribution of the stress in the Metal Framework for all angles values (**Table 3**), maximum value was 828 MPa for loading with angle of 15˚, about 48% higher than reference angle value 90˚.

4. DISCUSSION

This study on the location of the stress concentration area innovated when considering the non-linear contact interfaces, not considered in previous studies [18-21]. This method allowed to evaluate the stress distribution in the assembly of some components of prothesis (implant, screw, and metal framework).

The occlusal force is a factor that has a direct to implant longevity, according to the contact between teeth and implants creates a type of loading that is transmitted up to interfaces of the components, especially the bone-implant interface. In the perimplantar region the loss of natural bone is the consequence of excessive occlusal forces that exceed the interface's ability to absorb tension, the implant will fail [10, 14].

Some author [10, 14, 22] make a point of attention to the different loading cycles result in alternation between contact and separation of components of prothesis causing bone loss. The analysis by 3D finite element methods was used in this study, because it allows the consideration of real geometries and more realistic physical conditions to nonlinear phenomena considered in this study [6, 7, 23].

The masticatory loads simulated in this study were 100 N, representing an average for the study because there is a large variation of forces reported for patients with implant ranging from 0 to 400 N axial

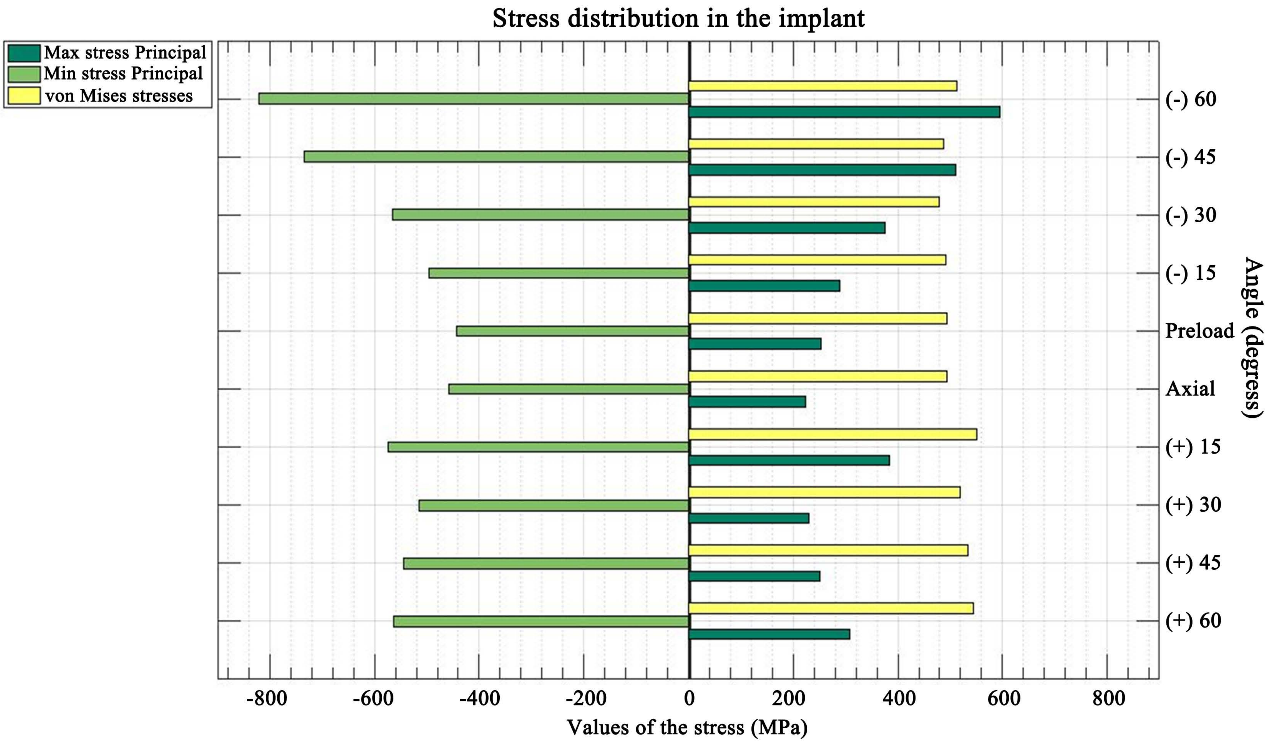

Figure 4. Comparison of stresses results for the implant. The maximum von Mises stress of implant and the peak minimum principal stress for different values of load angles.

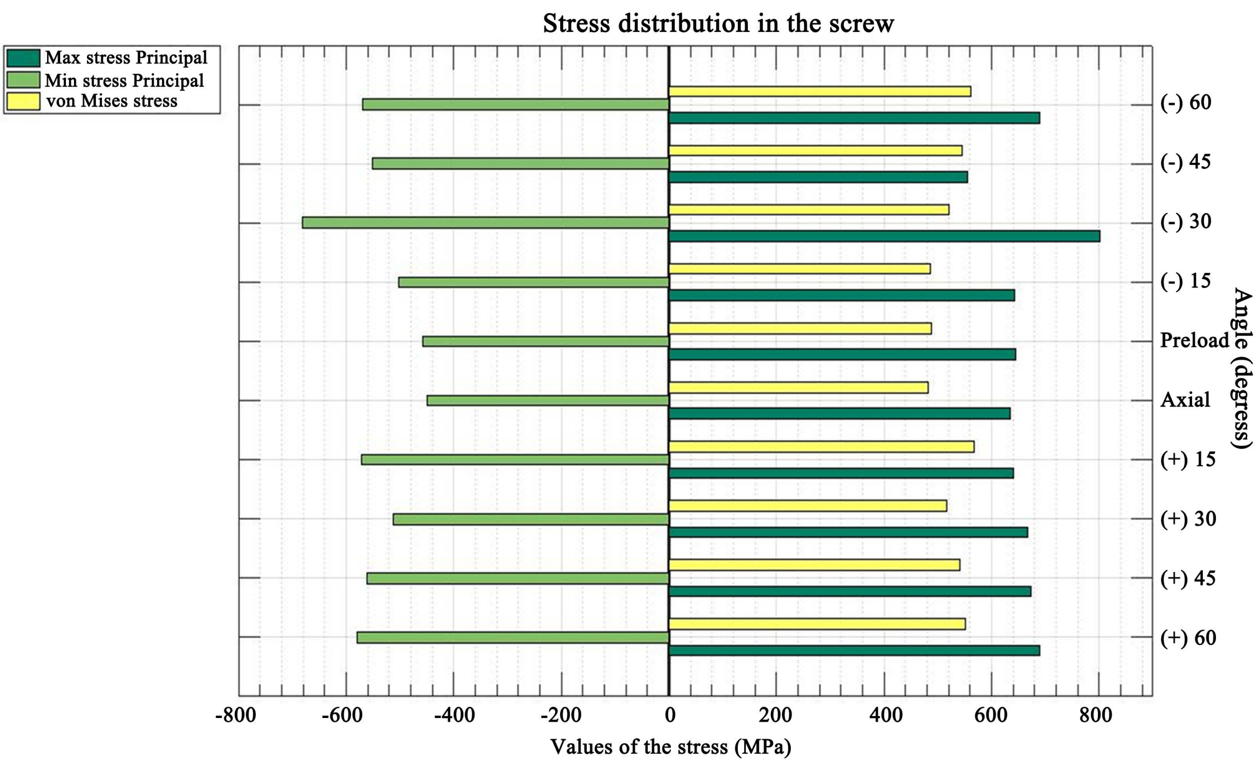

Figure 5. Comparison of stresses results for the screw. The maximum von Mises stress of implant and the peak minimum principal stress for different values of load angles.

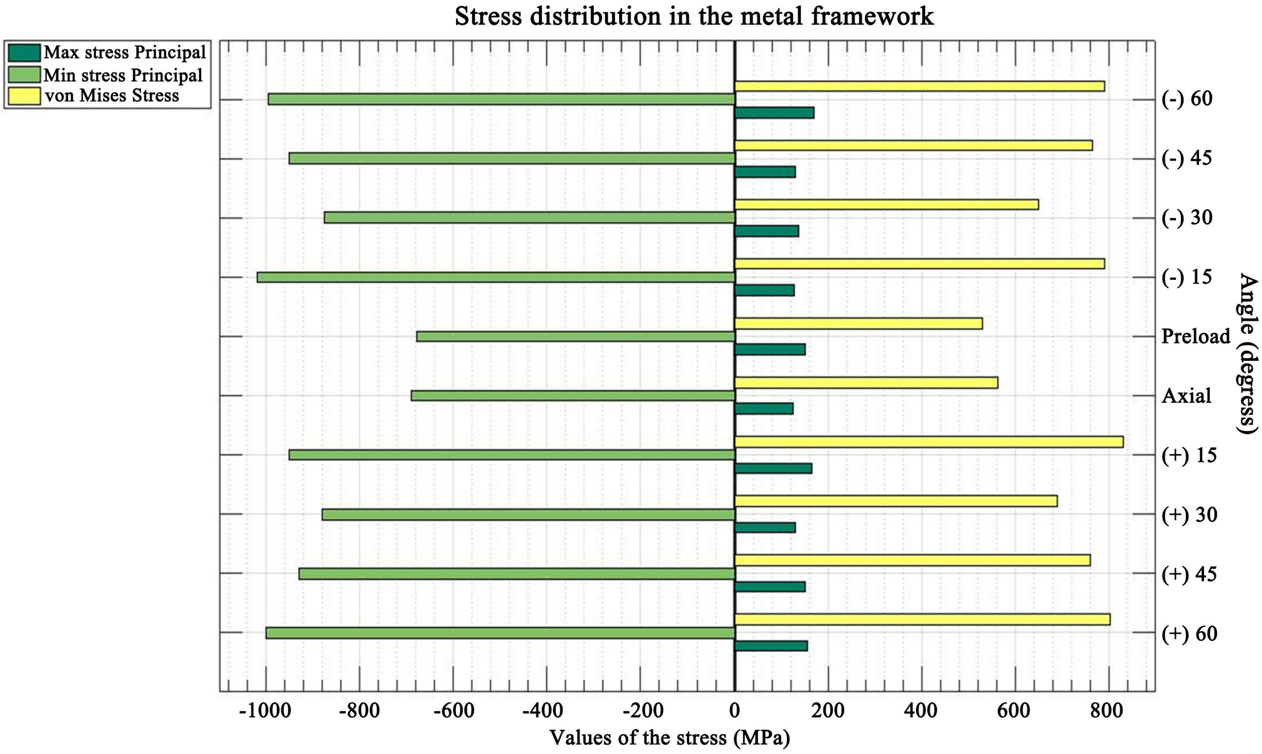

Figure 6. Comparison of stress results for metal framework. The maximum von Mises stress of implant and the peak minimum principal stress for different values of load angles.

force [13]. There are numerous reports of experiment evidence suggesting that ideal occlusal loading for analysis of the prothesis in the resultant should be placed 1 mm from axial of symmetry a 11.3˚ inclination [13, 24].

Analyzing the load application angle, a trend was observed of higher equivalent von Mises stress at an angle of 15˚ with higher concentration of stresses in the interface screw-implant in the first and second threads of the implant. This result confirm clinical studies, in which the most common complication found are in the screw, presenting loosening and/or fracture of the screw [25-27].

Our results showed that it possible to evaluate in detail the magnitude and tension in the interfaces studied for nonlinear boundary conditions considering the friction factor. In this study, static behaviors of the components of prothesis are investigated under conditions for different loading. For the conditions studied, the maximum stress values are lower than the material yield stress for the evaluated materials. It can be said that for the static analysis the implant is durable.

Finally, it can be affirmed the computational numerical model using the finite element method applied for the supported dental implant prothesis through the techniques described in this work and combined to computational simulation can be used as an acceptable predictor of life to the dental prothesis supported implant.

5. CONCLUSIONS

This study found that the interfaces assumption of the perfectly bonded to components of prothesis. This assumption does not appear realistic for the clinical condition.

To incorporate the effect of screw preload into the finite element model, the preload condition may be specified by the Equation (1) and a coefficient of friction of 0.3 between all contacting surfaces. In general, overloading occurs near the superior region of screw, and it is mainly caused by the normal and lateral components of the occlusal forces.

The assumption of fully bonded interfaces may lead to erroneous result. However, adding the friction in surfaces contacting may be recommend to dental to get better prothesis designs and contributing optimization of the implant.

CONFLICTS OF INTEREST

The authors declare no conflicts of interest regarding the publication of this paper.

REFERENCES

1. Wu, T., Fan, H., Ma, R., Chen, H., Li, Z. and Yu, H. (2017) Effect of Lubricant on the Reliability of Dental Implant Abutment Screw Joint: An *in Vitro* Laboratory and Three-Dimension Finite Element Analysis. *Materials Science and Engineering: C*, **75**, 297-304. http://linkinghub.elsevier.com/retrieve/pii/S092849311630892X https://doi.org/10.1016/j.msec.2016.11.041
2. Bulaqi, H.A., Mashhadi, M.M., Safari, H., Samandari, M.M., Geramipanah, F., Mousavi Mashhadi, M., *et al.* (2015) Effect of Increased Crown Height on Stress Distribution in Short Dental Implant Components and Their Surrounding Bone: A Finite Element Analysis. *The Journal of Prosthetic Dentistry*, **113**, 548-557. http://www.sciencedirect.com/science/article/pii/S0022391315000116 https://doi.org/10.1016/j.prosdent.2014.11.007
3. Xia, D., Lin, H., Yuan, S., Bai, W. and Zheng, G. (2014) Dynamic Fatigue Performance of Implant-Abutment Assemblies with Different Tightening Torque Values. *Bio-Medical Materials and Engineering*, **24**, 2143-2149. https://doi.org/10.3233/BME-141025
4. Khraisat, A., Hashimoto, A., Nomura, S. and Miyakawa, O. (2004) Effect of Lateral Cyclic Loading on Abutment Screw Loosening of an External Hexagon Implant System. *The Journal of Prosthetic Dentistry*, **91**, 326-334. http://www.sciencedirect.com/science/article/pii/S0022391304000022 https://doi.org/10.1016/j.prosdent.2004.01.001
5. Misch, C. (2015) Próteses sobre Implantes Dentais. Elsevier, Brasil.
6. Bulaqi, H.A., Mousavi Mashhadi, M., Geramipanah, F., Safari, H. and Paknejad, M. (2015) Effect of the Coefficient of Friction and Tightening Speed on the Preload Induced at the Dental Implant Complex with the Finite Element Method. *The Journal of Prosthetic Dentistry*, **113**, 405-411. http://www.sciencedirect.com/science/article/pii/S0022391314004806 https://doi.org/10.1016/j.prosdent.2014.09.021
7. de Cos Juez, F.J.J., Sánchez Lasheras, F., Garcia Nieto, P.J.J. and Alvarez-Arenal, A. (2008) Non-Linear Numerical Analysis of a Double-Threaded Titanium Alloy Dental Implant by FEM. *Applied Mathematics and Computation*, **206**, 952-967. http://www.sciencedirect.com/science/article/pii/S0096300308007674 https://doi.org/10.1016/j.amc.2008.10.019
8. Amorim Vasco, M.A.A., Doblaré Castellano, M., Bayod López, J. and Barbosa de las Casas, E. (2016) Utilização de tomografias computadorizadas de baixa resolução para construção de modelos geométricos detalhados de mandíbulas com e sem dentes. *Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería*, **32**, 1-6. http://www.sciencedirect.com/science/article/pii/S0213131514000625 https://doi.org/10.1016/j.rimni.2014.09.003
9. Misch, C.E. (2002) The Effect of Bruxism on Treatment Planning for Dental Implants. *Dentistry Today*, **21**, 76-81.
10. Wolff, J., Narra, N., Antalainen, A.K., Valásek, J., Kaiser, J., Sándor, G.K., *et al.* (2014) Finite Element Analysis of Bone Loss around Failing Implants. *Materials & Design*, **61**, 177-184. http://www.sciencedirect.com/science/article/pii/S0261306914003550 https://doi.org/10.1016/j.matdes.2014.04.080

11. Farina, A.P., Spazzin, A.O., Xediek Consani, R.L. and Mesquita, M.F. (2014) Screw Joint Stability after the Application of Retorque in Implant-Supported Dentures under Simulated Masticatory Conditions. *Journal of Prosthetic Dentistry*, **111**, 499-504. https://doi.org/10.1016/j.prosdent.2013.07.024
12. Kayabaşı, O., Yüzbasıoğlu, E. and Erzincanlı, F. (2006) Static, Dynamic and Fatigue Behaviors of Dental Implant Using Finite Element Method. *Advances in Engineering Software*, **37**, 649-658. http://www.sciencedirect.com/science/article/abs/pii/S0965997806000226 https://doi.org/10.1016/j.advengsoft.2006.02.004
13. Lin, D., Li, Q., Li, W. and Swain, M. (2009) Dental Implant Induced Bone Remodeling and Associated Algorithms. *Journal of the Mechanical Behavior of Biomedical Materials*, **2**, 410-432. http://www.sciencedirect.com/science/article/pii/S175161610800115X https://doi.org/10.1016/j.jmbbm.2008.11.007
14. Macedo, J.P., Pereira, J., Faria, J., Pereira, C.A., Alves, J.L., Henriques, B., *et al.* (2017) Finite Element Analysis of Stress Extent at Peri-Implant Bone Surrounding External Hexagon or Morse Taper Implants. *Journal of the Mechanical Behavior of Biomedical Materials*, **71**, 441-447. https://doi.org/10.1016/j.jmbbm.2017.03.011
15. Schwitalla, A.D., Abou-Emara, M., Spintig, T., Lackmann, J., Mueller, W.D., Muller, W.D., *et al.* (2015) Finite Element Analysis of the Biomechanical Effects of PEEK Dental Implants on the Peri-Implant Bone. *Journal of Biomechanics*, **48**, 1-7. http://www.sciencedirect.com/science/article/pii/S0021929014006058 https://doi.org/10.1016/j.jbiomech.2014.11.017
16. Cowin, S.C. (1989) The Mechanical Properties of Cortical Bone Tissue. CRC Press, Boca Raton.
17. Rice, J.C., Cowin, S.C. and Bowman, J.A. (1988) On the Dependence of the Elasticity and Strength of Cancellous Bone on Apparent Density. *Journal of Biomechanics*, **21**, 155-168. https://doi.org/10.1016/0021-9290(88)90008-5
18. Demenko, V., Linetskiy, I., Linetska, L. and Yefremov, O. (2019) Load-Carrying Capacity of Short Implants in Edentulous Posterior Maxilla: A Finite Element Study. *Medical Engineering Physics*, **71**, 30-37. https://doi.org/10.1016/j.medengphy.2019.02.003
19. Delgado-Ruiz, R.A., Calvo-Guirado, J.L. and Romanos, G.E. (2000) Effects of Occlusal Forces on the Peri-Implant-Bone Interface Stability. *Periodontology*, **81**, 179-193. https://doi.org/10.1111/prd.12291
20. Gupta, S., Chopra, P., Goyal, P. and Jain, A. (2021) Effect of Vertical Bone Loss on Stress Distribution at the Bone-Implant Interface around Implants of Varying Diameters an *in Silico* 3D Finite Element Analysis Is in Press. *Materials Today*: *Proceedings*, **28**, 1503-1509. https://doi.org/10.1016/j.matpr.2021.01.008
21. Demenko, V., Linetskiy, I., Nesvit, K. and Shevchenko, A. (2011) Ultimate Masticatory Force as a Criterion in Implant Selection. *Journal of Dental Research*, **90**, 1211-1215. https://doi.org/10.1177/0022034511417442
22. Ali, B., Ould Chikh, E.B., Meddah, H.M., Merdji, A. and abbes Bachir Bouiadjra, B. (2013) Effects of Overloading in Mastication on the Mechanical Behaviour of Dental Implants. *Materials & Design*, **47**, 210-217. http://www.sciencedirect.com/science/article/pii/S0261306912008400 https://doi.org/10.1016/j.matdes.2012.12.019
23. Huang, H.L., Hsu, J.T., Fuh, L.J., Tu, M.G., Ko, C.C. and Shen, Y.W. (2008) Imaging Finite Element Analysis Cancellous Bone Dental Implants Reconstruction Grafts Dentistry, Oral Surgery & Medicine. *Journal of Dentistry*, **36**, 409-417. https://doi.org/10.1016/j.jdent.2008.02.015
24. Bozkaya, D., Muftu, S. and Muftu, A. (2004) Evaluation of Load Transfer Characteristics of Five Different Implants in Compact Bone at Different Load Levels by Finite Elements Analysis. *The Journal of Prosthetic Dentistry*, **92**, 523-530. http://www.thejpd.org/article/S0022-3913(04)00490-1/pdf https://doi.org/10.1016/j.prosdent.2004.07.024
25. Shemtov-Yona, K., Rittel, D., Levin, L. and Machtei, E.E. (2014) Effect of Dental Implant Diameter on Fatigue

Performance. Part I: Mechanical Behavior. *Clinical Implant Dentistry and Related Research*, **16**, 172-177. https://doi.org/10.1111/j.1708-8208.2012.00477.x

26. Shemtov-Yona, K. and Rittel, D. (2015) On the Mechanical Integrity of Retrieved Dental Implants. *Journal of the Mechanical Behavior of Biomedical Materials*, **49**, 290-299. http://www.sciencedirect.com/science/article/pii/S1751616115001812 https://doi.org/10.1016/j.jmbbm.2015.05.014
27. Mohammed, H.H., Lee, J.H., Bae, J.M. and Cho, H.W. (2016) Effect of Abutment Screw Length and Cyclic Loading on Removal Torque in External and Internal Hex Implants. *The Journal of Advanced Prosthodontics*, **8**, 62-69. http://www.ncbi.nlm.nih.gov/pmc/articles/PMC4769891 https://doi.org/10.4047/jap.2016.8.1.62