Published online: 2019-09-26

A three-dimensional finite element study on the stress distribution pattern of two prosthetic abutments for external hexagon implants

Wagner Moreira[1,2], Caio Hermann[1], Jucélio Tomás Pereira[3], Jean Anacleto Balbinoti[3], Rodrigo Tiossi[1,4]

[1]Department of Prosthodontics, Latin American Institute of Dental Research and Education, ILAPEO, Curitiba, Paraná, Brazil,
[2]PhD Student, São Leopoldo Mandic College, Campinas, São Paulo, Brazil,
[3]Department of Mechanical Engineering, Federal University of Paraná, Curitiba, Paraná, Brazil,
[4]Department of Odontotécnica, Niterói School of Dentistry, Fluminense Federal University, Niterói, Rio de Janeiro, Brazil

Correspondence: Dr. Rodrigo Tiossi
Email: rtiossi@yahoo.com

ABSTRACT

Objective: The purpose of this study was to evaluate the mechanical behavior of two different straight prosthetic abutments (one- and two-piece) for external hex butt-joint connection implants using three-dimensional finite element analysis (3D-FEA). **Materials and Methods:** Two 3D-FEA models were designed, one for the two-piece prosthetic abutment (2 mm in height, two-piece mini-conical abutment, Neodent) and another one for the one-piece abutment (2 mm in height, Slim Fit one-piece mini-conical abutment, Neodent), with their corresponding screws and implants (Titamax Ti, 3.75 diameter by 13 mm in length, Neodent). The model simulated the single restoration of a lower premolar using data from a computerized tomography of a mandible. The preload (20 N) after torque application for installation of the abutment and an occlusal loading were simulated. The occlusal load was simulated using average physiological bite force and direction (114.6 N in the axial direction, 17.1 N in the lingual direction and 23.4 N toward the mesial at an angle of 75° to the occlusal plan). **Results:** The regions with the highest von Mises stress results were at the bottom of the initial two threads of both prosthetic abutments that were tested. The one-piece prosthetic abutment presented a more homogeneous behavior of stress distribution when compared with the two-piece abutment. **Conclusions:** Under the simulated chewing loads, the von Mises stresses for both tested prosthetic-abutments were within the tensile strength values of the materials analyzed which thus supports the clinical use of both prosthetic abutments.

Key words: Dental implant-abutment interface, dental implants, finite element analysis, platform switching

INTRODUCTION

Implant dentistry initially aimed to restore fully edentulous arches using implant-fixed complete dentures.[1] With the high success rates that followed, the principles of the implant treatment were applied in the restoration of partially edentulous patients.[2] The primary treatment objective is the re-establishment of function.[3] Further, objectives include the long-term functional stability of the implants, reduced surgical and prosthetic procedures, high predictability of the treatment outcomes, and optimal framework design.[3]

The transference of the occlusal forces to the bone-implant interface is a crucial factor to determine the outcome of the implant treatment.[4] It is therefore essential an implant design capable to distribute the functional forces to the supporting

How to cite this article: Moreira W, Hermann C, Pereira JT, Balbinoti JA, Tiossi R. A three - dimensional finite element study on the stress distribution pattern of two prosthetic abutments for external hexagon implants. Eur J Dent 2013;7:484-91.

 DOI: 10.4103/1305-7456.120642

structures within physiological values.[4] The design of the interface connection between the implant head and the prosthetic abutment is one of the differences between the commercially available implant-systems that can affect the biomechanical behavior of the implants.[4,5] Among the popular designs for abutment connections are the internal and external hexagons and the internal conical.[4-7] The implant abutment connection can influence the loosening and/or the fracture of the abutment screw as well as how the forces are transferred to the implant-bone interface[6] and to the implant-prosthetic abutment interface.[7] Joint strength and stability, the mechanical integrity of the implant-abutment complex, and the force magnitudes near the implants are determined by the design of the implant-abutment interface.[5,7,8]

The preservation of crestal bone levels around the cervical region of implants using the concept of platform switching has been previously described and found satisfactory results.[9] The installation of smaller diameter prosthetic abutments in implants with 5.0 and 6.0 diameter has demonstrated a smaller than expected vertical change in the crestal bone height around implants with external hex butt-joint connections.[9] However, when 4.1 mm diameter external hex implants are used, a prosthetic component of matching diameter is needed. This has led to the development of a prosthetic component for external hex implants with 4.1 mm in diameter but with a narrow emergence profile.

There are therefore two straight prosthetic abutments for screw-retained prosthesis supported by implants with an external hex connection: The standard solid abutment with its retaining screw as an extension of the abutment itself that can also be defined as a one-piece abutment; the other is a two-piece abutment, with a separate independent screw that matches its counterpart in the implant body.[8] Another design feature of the one-piece abutment is the narrower emergence profile when compared to the two-piece abutment.

The preload levels achieved by the abutments play a crucial role in the maintenance of the implant-abutment interface.[8] The pattern of stress distribution and the biomechanical behavior of the different prosthetic abutments that were previously described and are currently available for external hex implant connections is yet not well-documented. Finite element analysis is a largely used and efficient technique for the evaluation of stress distribution patterns at the bone-implant interface as well as at the implant-abutment interface. With the use of finite element modeling, this study aims to compare the preload levels after torque application for the installation of the two different straight prosthetic abutments (one- and two-piece) and the pattern of stress distribution after simulating an occlusal load on the same abutments. The null hypothesis was that no differences would be found between the two tested prosthetic abutments and that the biomechanical behavior of the two prosthetic abutments would be similar.

MATERIALS AND METHODS

A cross-section of a volumetric cone-beam computed tomography (CT) (Galileos, SIRONA Dental Systems GmbH, Bensheim, Hesse, Germany) of the first premolar region was used to create a computer-aided design (CAD) model of an edentulous mandible. Specialized computer software (Dental Slice 2.7.2, Bioparts, Brasília, DF, Brazil) was used to design the model of the mandible using the coordinates from the CT images of the mandible of the patient (DYCON), allowing adequate shape, thickness, and amount of cortical and cancellous bone.

The outlined model was transferred to a CAD software (SolidWorks 2007, SolidWorks Corporation, Santa Monica, CA, USA) to simulate a three-dimensional model of a dry human skull with 8 mm in mesio-distal length for each side of the section, exceeding the minimum length of 4.2 mm as previously recommended,[10] and radius of curvature of 33.5 mm. The alveolar ridge was 6.5 mm long labiolingually and a uniform 1-mm-thick layer of cortical bone was modeled on the buccal and lingual aspects.[11] Soft- tissues such as the inferior alveolar nerve, periodontal ligament, and pulp were not modeled due to their limited visibility in CT images.[12] Certain assumptions regarding material properties and boundary conditions were needed to make the modeling and solving process possible.[13] A distance of 0.005 mm between the contacting elements in finite element models was assumed.[13] In addition, a coefficient of friction of 0.3 between the contacted surfaces was used based on values from the literature.[13,14]

A cylindric external hex implant (3.75 mm in diameter and 13 mm in length, Titamax Ti Cortical, Neodent, Curitiba, PR, Brazil) was placed in the middle of the simulated mandible. For this study, two similar

3D finite element (FE) models were simulated, one with the two-piece straight prosthetic abutment (4.1 mm in diameter and 2 mm in height, mini conical abutment, Neodent) (M1), and another one with the one-piece straight prosthetic abutment (4.1 mm in diameter and 2 mm in height, Slim Fit® mini conical abutment, Neodent) (M2). The company that manufacturers the implants and implant-components provided the CAD images of the materials used in this comparative study (Neodent).

To simplify the computation processes, all materials were considered as isotropic, homogeneous, and linearly elastic. Material properties were collected from relevant literature [Table 1].[10,11] The 3D-FE models and the properties of the bone structure and materials were exported to the FE software (Ansys Workbench 10, Swanson Analysis Systems Inc., Houston, PA, USA) to run the simulations. The characteristics of the constructed models were: M1, 234,688 elements and 383,547 nodes; M2, 233,754 elements and 379,949 nodes [Figure 1a and b].

The loadings for this study were applied in two steps: Preload after torque application for installation of the abutment (t = 1 s) and occlusal loading (t = 2 s). The preload condition was achieved by the use of contact analysis in the FE models.[13] To simulate the preload condition, the target and contact surfaces between the individual parts of the model were defined by not merging the nodes between the components.[13] According to settings from a previous study,[13] contact analysis assured the union and the transfer of the loads and deformation between the different components, featuring a coefficient of friction of 0.3. A 20 N-cm torque was used for the installation of the prosthetic abutments as recommended by the manufacturer.

The occlusal loading force applied to the prosthetic abutments was a combination of 114.6 N in the axial direction, 17.1 N in the lingual direction and 23.4 N toward the mesial at an angle of about 75° to the occlusal plan.[15] The mandible was considered a fixed structure without freedom of movement and completely bonded to the implants (osseointegrated in perfect condition).[6] All movements were restricted in all directions during load application and the boundary conditions considered the outer surfaces of the geometric model in the mesio-distal direction as fixed. The von Mises stress values were used to compare the two models analyzed in this study.

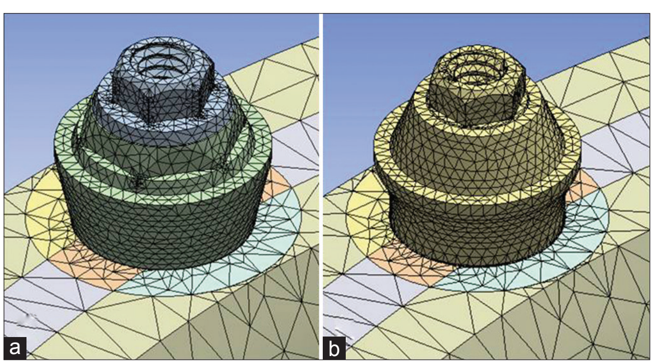

Figure 1: (a) Finite element-mesh generated for M1. (b) FE-mesh generated for M2

RESULTS

When the preload on the abutment screws was simulated, no stresses were transferred to the bone tissues surrounding the implants in both groups. After load application, both groups transferred von Mises stress values of 80 MPa to the surrounding bone structures. The von Mises stress results found for the two prosthetic abutments tested in this study are presented in Tables 2 and 3. Figures 2-8 show the

Table 1: Mechanical properties for the materials used in the present study

Materials	Modulus of elasticity (GPa)	Poisson ratio
Cortical bone	13	0.3
Cancellous bone	1.6	0.3
Implants (Ti-GR4)	105	0.37
Prosthetic abutments (Ti-6Al-4V)	110	0.34

Table 2: von Mises stress values (MPa) found the two-piece abutment

Regions in the abutment	Simulation of preload	Occlusal load
Head of the screw	874	0
Screw body	100	80
Body-head screw transition	210	160
Initial threads of the abutment screw	280	315
Implant	135	170
Implant-abutment interface	110	125

Table 3: von Mises stress values (MPa) found the one-piece abutment

Regions in the abutment	Simulation of preload	Occlusal load
Head of the screw	Near zero stresses	0
Screw body	105	70
Body-head screw transition	150	125
Initial threads of the abutment screw	220	230
Implant	150	170
Implant-abutment interface	110	125

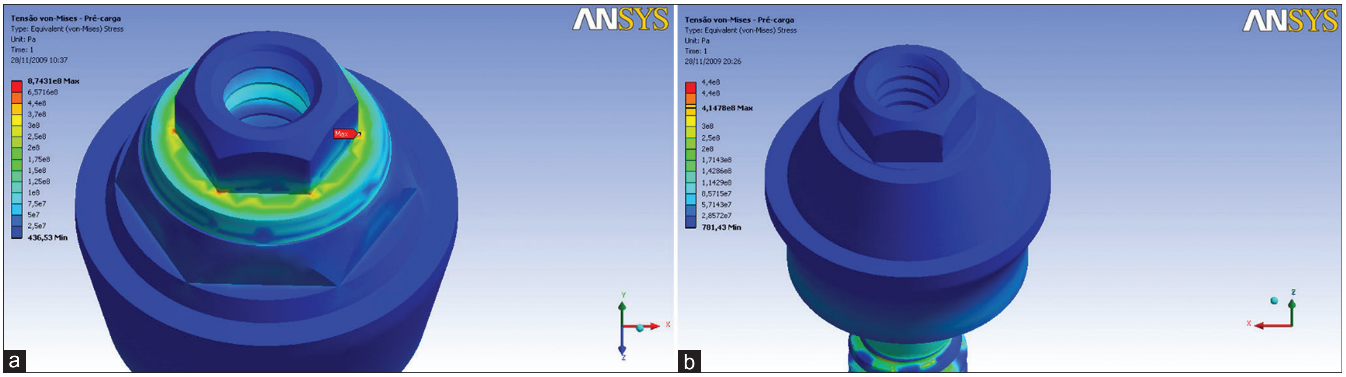

Figure 2: Preload stresses after torque application (A: M1, B: M2)

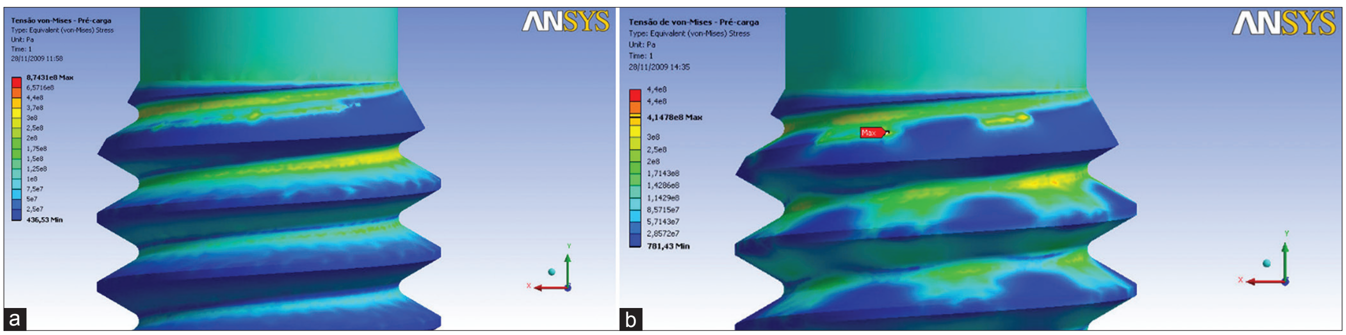

Figure 3: Stresses on the initial threads of the screws for both models after preload (A: M1, B: M2)

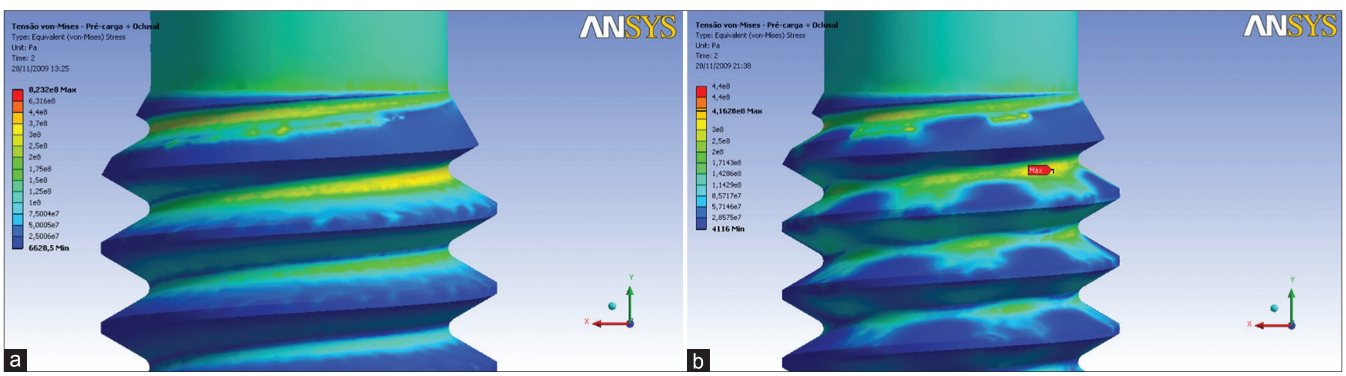

Figure 4: Stresses on the initial threads of the screws for both models after occlusal loading of the models (A: M1, B: M2)

stress pattern distribution for the groups that were analyzed.

When the two-piece prosthetic abutment (M1) was screwed to the implant, an 874 MPa stress on the head of the screw was found [Figure 2a] caused by the preload of the screw. Conversely, when preload was applied to the screw of the one-piece abutment (M2), no stresses were found at the head of the screw [Figure 2b]. The highest von Mises stress values (280 MPa) found on the screw of the two-piece abutment were at the first two threads [Figure 3a], indicating that this could be the best region of the screw to evaluate the influence of the preload on the screws. The stresses in the first two threads of the screw in the one-piece abutment (220 MPa) were lower than that in the same region of the two-piece abutment (280 MPa) [Figure 3b]. Under occlusal loading, the two-piece abutment presented increased von Mises stress values (315 MPa) at the first two threads of the abutment screw [Figure 4a]. The one-piece abutment also had increased stresses in the same region (230 MPa) [Figure 4b].

The preload stresses in the region of the screw body were similar for both prosthetic abutments analyzed (M1: 100 MPa; M2: 105 MPa). For the two-piece abutment, the stresses in the transition between the body and the head of the screw (210 MPa) were higher than in the one-piece abutment (150 MPa). Under occlusal load, the two tested abutments presented

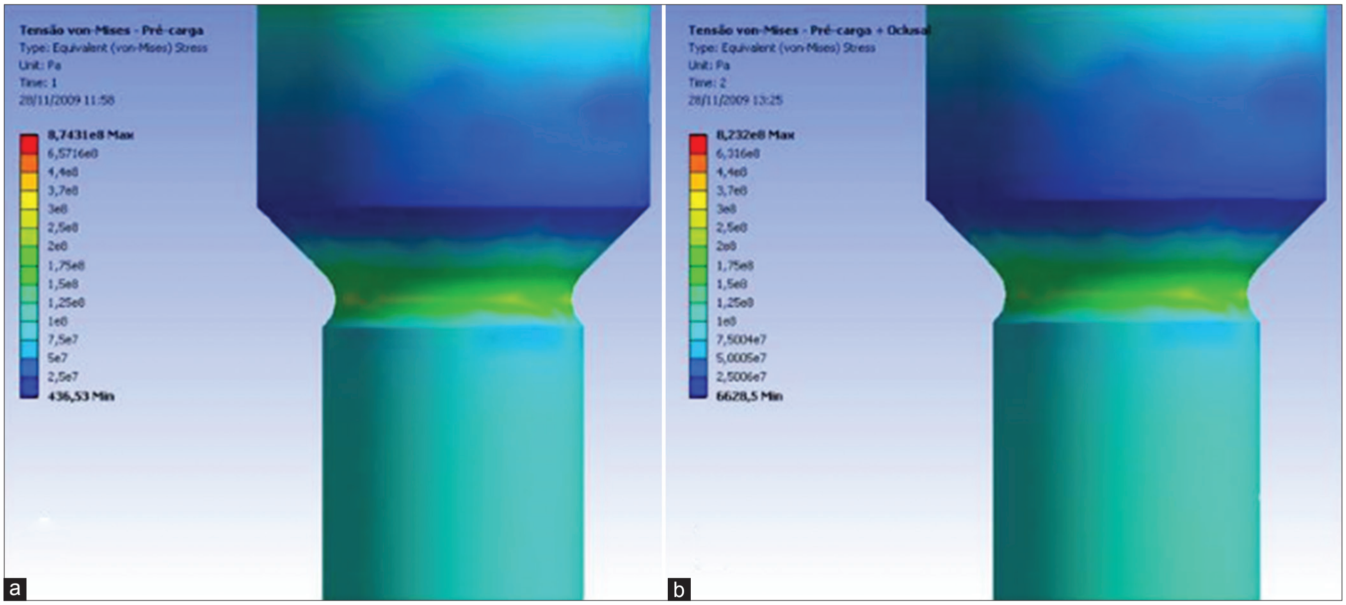

Figure 5: Two-piece prosthetic abutment (regions: Body and body-head screw transition). (a) Stresses found after simulation of the preload. (b) Stresses found after simulation of the occlusal loadings

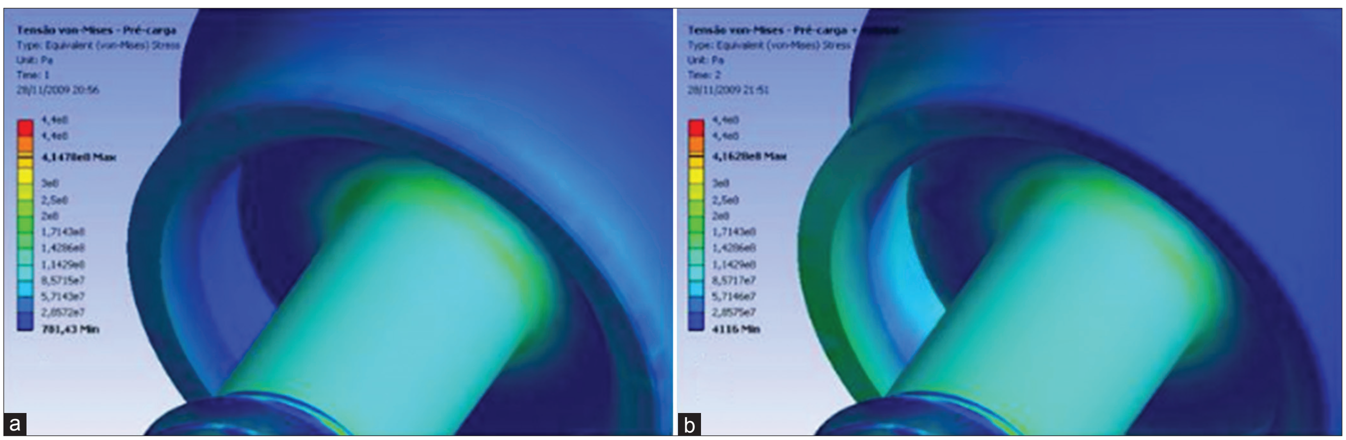

Figure 6: One-piece prosthetic abutment (regions: Body and body-head screw transition). (a) Stresses found after simulation of the preload. (b) Stresses found after simulation of the occlusal loadings

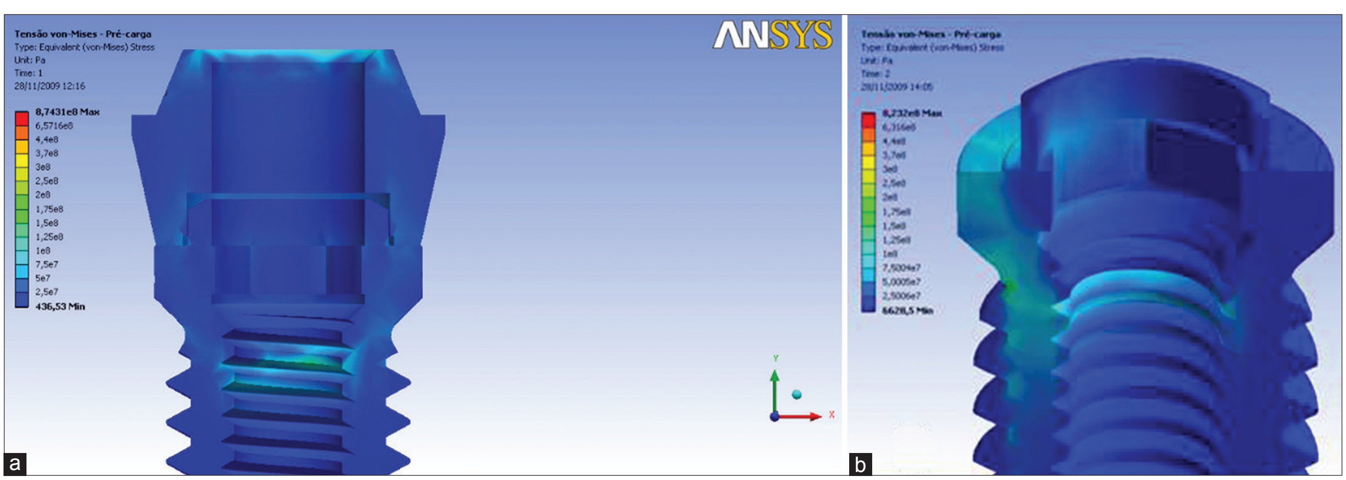

Figure 7: Stresses found on the implant wall and on the implant/abutment interface for the two-piece abutment. (a) Simulation of preload. (b) After occlusal loading

a reduction of the stresses at the body of the screw (M1, 80 MPa, and M2, 70 MPa) and at the transition between the body and the head of the screw (M1, 160 MPa, and M2, 125 MPa) [Figures 5 and 6].

When compared to the preload in the two-piece abutment, the occlusal loading increased the stresses at the implant/abutment interface from 135 MPa to 170 MPa at the implant wall and from 110 MPa to 125 MPa at the abutment [Figure 7a and b]. Similar results and stress distribution were found for the one-piece abutment, with the difference that the stresses at the implant wall for this abutment were higher after preload application (150 MPa) [Figure 8a and b].

The tensile strength values for each material were collected from the literature[16,17] and compared to the highest von Mises stresses that were found for each FE model, aiming to understand whether the prosthetic components could tolerate the mechanical stresses during functional loading. The results are presented in Table 4.

Table 4: Highest von Mises stress values (MPa) and tensile strength (MPa) for each component evaluated

Models	Component	von Mises stress	Tensile strength
I	Two-piece abutment	315	860
I	Implant	170	550
II	One-piece abutment	230	860
II	Implant	170	550

DISCUSSION

This study evaluated the stress distribution in one- and two-piece straight prosthetic abutments for implant-supported prosthesis. The influence of the preload caused by tightening the screws for abutment installation and the stresses transferred to the implants and implant components after load application were evaluated. The results support acceptance of the tested null hypothesis as there were no differences between the two tested prosthetic abutments. However, the one-piece mini-conical abutment (M2) presented a more homogeneous behavior of stress distribution [Figures 5 and 6].

The amount of stresses (80 MPa) transferred to the surrounding bone structures after applying the loads on the models are in agreement with previous studies.[18-20] The lower stress values found in the screw threads for the one-piece abutment can be due to the higher stresses in the abutment body; this thus relieves the stresses in the screws. Previous studies that compared one- and two-piece prosthetic abutments also found minimized stresses in the screws of one-piece abutments.[21,22] However, the afore-mentioned studies evaluated internal Morse-taper connections instead of external hex butt-joint configurations.[21,22]

For the two-piece abutment, the stresses in the transition between the body and the head of the

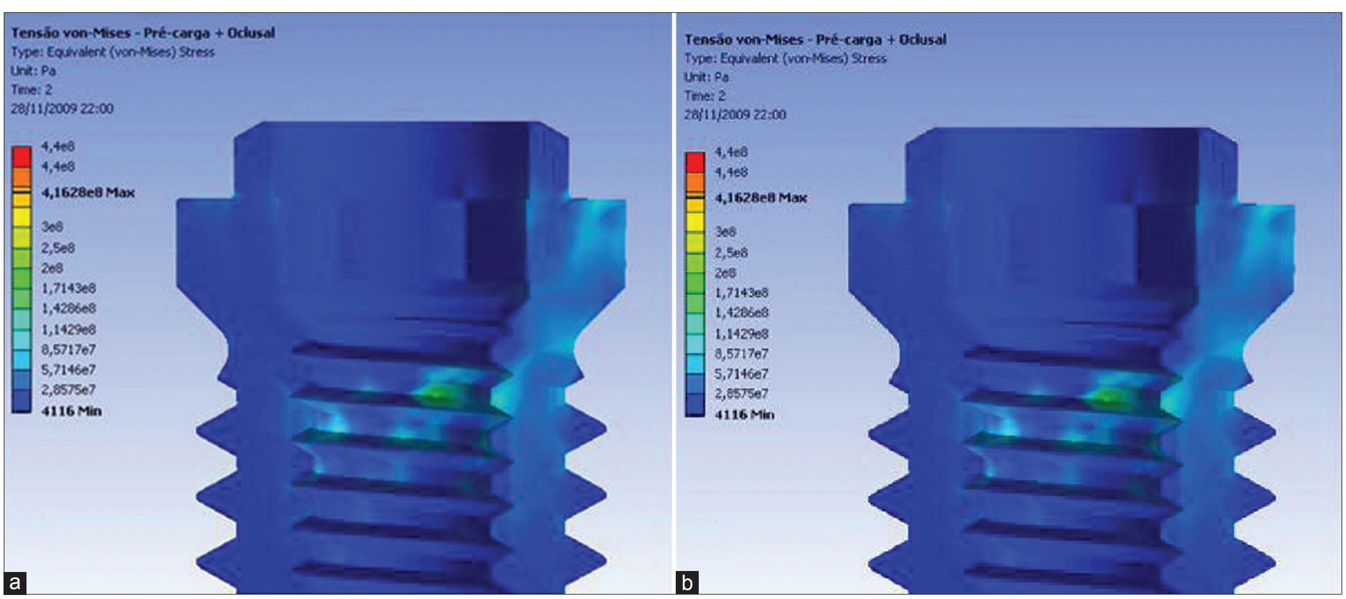

Figure 8: Stresses found on the implant wall and on the implant/abutment interface for the one-piece abutment. (a) Simulation of preload. (b) After occlusal loading

screw (210 MPa) were higher than in the one-piece abutment (150 MPa). It can be speculated that a decrease in the diameter of the screw in the body-to-head of the screw transition might concentrate the stresses in this small region. A previously published FE study found that for every 1.0 µm elongation of the screw would be equivalent to a 47.9 N increase of the preload in the implant complex.[23] Under occlusal load, the two tested abutments presented a reduction of the stresses at the body of the screw (M1, 80 MPa, and M2, 70 MPa) and at the transition between the body and the head of the screw (M1, 160 MPa, and M2, 125 MPa).

In the implants and in the implant/abutment interface, both groups presented the same von Mises stresses after the simulated occlusal loads, suggesting that regardless of the abutment type, the stresses in the implants are the same. In addition, higher mechanical stresses are expected near the screw head of two-piece prosthetic abutments[24] and implants[18,25] under occlusal loading. The von Mises stresses under the simulated chewing loads were all within the tensile strength of the materials analyzed, which thus validates the clinical use of both prosthetic abutments. The narrower emergence profile of the one-piece abutment [Figure 1b] compared to the two-piece abutment could allow a more subcrestal placement of external-hex implants. However, it cannot be stated that the new design applies the concept of platform switching. Instead of having a smaller diameter than the implant, the abutment presents a narrower emergence profile than the conventional abutments for external hex implants. The latter usually presents a more convex and wider emergence profile [Figure 1a].

The FE method has been widely used for biomechanical analysis of human joints and implants.[26,27] Due to limited computing power and resources, a specific region of interest is normally selected for 3D analysis to allow analysis to be performed on a more detailed and complex structure.[26] According to settings from a previous study,[28] three consecutive iterations of mesh refinement were performed in each model to observe the convergence of the results. The assumptions regarding material properties and boundary conditions that were needed for this study should be taken into account when analyzing the results that were found. The effects of dynamic loading and the clinical behavior of the tested prosthetic abutments therefore require further investigation.

CONCLUSIONS

Based on the results found in this study and within the limitations of the methodology that was used, it can be concluded that:

- The one-piece mini-conical abutment (M2) presented a more homogeneous behavior of stress distribution
- Within the testing conditions used in this study, no plastic deformation of the implants or implant-components is expected for both prosthetic abutments that were tested.

ACKNOWLEDGMENTS

The authors wish to thank NEODENT for supplying the computer-aided design images for the implants and prosthetic abutments.

REFERENCES

1. Adell R, Lekholm U, Rockler B, Brånemark PI. A 15-year study of osseointegrated implants in the treatment of the edentulous jaw. Int J Oral Surg 1981;10:387-416.
2. Balshi TJ, Hernandez RE, Pryszlak MC, Rangert B. A comparative study of one implant versus two replacing a single molar. Int J Oral Maxillofac Implants 1996;11:372-8.
3. Buser D, Belser UC, Lang NP. The original one-stage dental implant system and its clinical application. Periodontol 2000 1998;17:106-18.
4. Cehreli M, Duyck J, De Cooman M, Puers R, Naert I. Implant design and interface force transfer. A photoelastic and strain-gauge analysis. Clin Oral Implants Res 2004;15:249-57.
5. Brunski JB. Biomaterials and biomechanics in dental implant design. Int J Oral Maxillofac Implants 1988;3:85-97.
6. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent 2001;85:585-98.
7. Akça K, Cehreli MC, Iplikçioğlu H. Evaluation of the mechanical characteristics of the implant-abutment complex of a reduced-diameter morse-taper implant. A nonlinear finite element stress analysis. Clin Oral Implants Res 2003;14:444-54.
8. Cehreli MC, Akça K, Iplikçioğlu H, Sahin S. Dynamic fatigue resistance of implant-abutment junction in an internally notched morse-taper oral implant: Influence of abutment design. Clin Oral Implants Res 2004;15:459-65.
9. Lazzara RJ, Porter SS. Platform switching: A new concept in implant dentistry for controlling postrestorative crestal bone levels. Int J Periodontics Restorative Dent 2006;26:9-17.
10. Teixeira ER, Sato Y, Akagawa Y, Shindoi N. A comparative evaluation of mandibular finite element models with different lengths and elements for implant biomechanics. J Oral Rehabil 1998;25:299-303.
11. Kao HC, Gung YW, Chung TF, Hsu ML. The influence of abutment angulation on micromotion level for immediately loaded dental implants: A 3-D finite element analysis. Int J Oral Maxillofac Implants 2008;23:623-30.
12. Zachrisson H, Engström E, Engvall J, Wigström L, Smedby O, Persson A. Soft tissue discrimination *ex vivo* by dual energy computed tomography. Eur J Radiol 2010;75:e124-8.
13. Alkan I, Sertgöz A, Ekici B. Influence of occlusal forces on stress distribution in preloaded dental implant screws. J Prosthet Dent 2004;91:319-25.
14. Parmley RO. Standard Handbook of Fastening and Joining. New York: McGraw-Hill; 1997.
15. Bozkaya D, Muftu S, Muftu A. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis. J Prosthet Dent

2004;92:523-30.
16. Niinomi M. Mechanical properties of biomedical titanium alloys. Mater Sci Eng A 1998;243:231-6.
17. Geetha M, Singh AK, Asokamani R, Gogia AK. Ti based biomaterials, the ultimate choice for orthopaedic implants: A review. Prog Mater Sci 2009;54:397-425.
18. Chun HJ, Shin HS, Han CH, Lee SH. Influence of implant abutment type on stress distribution in bone under various loading conditions using finite element analysis. Int J Oral Maxillofac Implants 2006;21:195-202.
19. Himmlová L, Dostálová T, Kácovský A, Konvicková S. Influence of implant length and diameter on stress distribution: A finite element analysis. J Prosthet Dent 2004;91:20-5.
20. Kong L, Hu K, Li D, Song Y, Yang J, Wu Z, *et al*. Evaluation of the cylinder implant thread height and width: A 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 2008;23:65-74.
21. Quaresma SE, Cury PR, Sendyk WR, Sendyk C. A finite element analysis of two different dental implants: Stress distribution in the prosthesis, abutment, implant, and supporting bone. J Oral Implantol 2008;34:1-6.
22. Pessoa RS, Muraru L, Júnior EM, Vaz LG, Sloten JV, Duyck J, *et al*. Influence of implant connection type on the biomechanical environment of immediately placed implants-CT-based nonlinear, three-dimensional finite element analysis. Clin Implant Dent Relat Res 2010;12:219-34.
23. Wang RF, Kang B, Lang LA, Razzoog ME. The dynamic natures of implant loading. J Prosthet Dent 2009;101:359-71.
24. Cehreli MC, Akça K, Iplikçioğlu H. Force transmission of one-and two-piece morse-taper oral implants: A nonlinear finite element analysis. Clin Oral Implants Res 2004;15:481-9.
25. Lan TH, Huang HL, Wu JH, Lee HE, Wang CH. Stress analysis of different angulations of implant installation: The finite element method. Kaohsiung J Med Sci 2008;24:138-43.
26. Saidin S, Abdul Kadir MR, Sulaiman E, Abu Kasim NH. Effects of different implant-abutment connections on micromotion and stress distribution: Prediction of microgap formation. J Dent 2012;40:467-74.
27. 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;91:144-50.
28. Saab XE, Griggs JA, Powers JM, Engelmeier RL. Effect of abutment angulation on the strain on the bone around an implant in the anterior maxilla: A finite element study. J Prosthet Dent 2007;97:85-92.

Access this article online	
Quick Response Code:	**Website:** www.eurjdent.com
	Source of Support: Nil. **Conflict of Interest:** None declared