Influence of external geometry of Morse dental implant on stress distribution

Roberto Brunow **LEHMANN***
Carlos Nelson **ELIAS****
Marco Aurélio **ZUCARELI*****

Abstract

Objective: To evaluate two proposals of external geometry of dental implants observing the influence on the stress distribution. **Methods:** It was performed the evaluation by finite elements of prototypes of dental implants with different external geometric shapes submitted to different conditions of loading (axial, inclined to 15° and inclined to 30°). **Results:** The stress increased as the loading became more inclined. The conical geometry showed itself more stable and transmitted less stress to the bone. **Conclusions:** I) The system with conical dental implant transmits lower stress to the bone and to the dental implant; II) the safety factor of the implants is high suggesting it supports loadings more aggressive in intensity and direction; III) as the loading becomes more inclined, i.e., the components of the lateral forces increase, the stresses on the bone and on the prosthetic components increase; IV) for all simulations, the systems behaved appropriately so there is no indication of deformation or fracture on the prosthetic components or even bone resorption due to overload.

Keywords: Computer simulation. Dental implant. Finite element analysis.

How to mention this article: Lehmann RB, Elias CN, Zucareli MA. Influence of external geometry of Morse dental implant on stress distribution. Dental Press Implantol. 2012 Apr-June;6(2):35-43.

» The authors inform that they do not have no associative, commercial, intellectual property or financial interests representing a conflict of interest, in products and companies described in this article.

Contact address

Roberto Brunow Lehmann
Rua Lourenço Leal, nº 77– Tambasco
Zip Code: 27.700-000 – Vassouras/RJ – Brazil
Email: rbrunow@gmail.com

Submitted on: 12/19/2011
Reviewed and received on: 03/06/2012

---

* Post-Doctorate in Development of Implants and Components, Laboratory of Computational Biomechanics. Federal Fluminense University.

** MSc and PhD in Sciences of Materials, Laboratory of Biomaterials. Militar Institute of Engineering – IME-RJ.

*** Engineer of materials, Engineering of products – Development of Implants and Components, Conexão Sistemas de Prótese.

Introduction

The Finite Elements Method (FEM) is regularly used for stress analysis in works of Dentistry.[1,2,3] Through computational tools it is possible to foresee the error as well as its location from the conditions of contour imposed on the model. The facility for obtention of results as well as its promptitude makes the FEM interesting in several departments as Dentistry, Implantology and Periodontics. There are works in literature[4,5,6] that evaluate the geometric variation of the screw threads of the implant. It was observed significant differences on the stress levels with the geometric variation. The results show that implants with slight curvature and lower depth threads present lower stress level[4] and there is greater concentration of stress on the first thread of the screw.[5,6] From the mechanical point of view, the analysis of the efforts transmitted on the interface bone-implant is essential to foresee the success of the osseointegrable dental implants. The overload may cause bone resorption or even failure.[7,8,9] On the other hand, a low intensity load may cause atrophy and subsequent bone loss.[7,9] The higher levels of bone stress are located on the marginal region of the implant, being considerate a critical region.[1,2,3] In the present study it was performed the analysis by finite elements through the three-dimensional models of dental implants prototypes [AR-Torq and Flash] and with variation on the direction of the equivalent loading to observe the behavior of both systems. Each one of these systems has different characteristics on its external geometry of the dental implant and, through the variation of loading, it is intended to evaluate the mechanical behavior of both systems.

Material and Methods

For the performance of the simulations it was used a microcomputer with Pentium Quad Core 6550 processor, RAM memory of 8GB and hard drive of 1,75TB. All dimensions used for the dental implants and prosthetic components were provided by Conexão Sistemas de Prótese (Arujá, Brazil) and refer to the prototypes that were in development with trade name AR Torq and Flash. Its characteristics can be followed on Table 1. The implant Ar-Torq is cylindrical self-threading, has indexed Morse, double screw and conical apex (Fig 1A). The Flash implant has cone Morse indexed and it is conical, has progressive thread with high cutting power and according to the manufacturer, it increases the superficial area and primary stability in relation to conventional implants (Fig 1B). Normally, the wall thickness of implants with internal connections limits the insertion torque and the axial loads supported are lower. The diameters of the evaluated implants were of 5 mm and length of 10 mm.

The cortical and medullary bones were modeled on the program ANSYS Workbench, version 11. The simplified dimensions of the bone were the available in literature[10-13]. Each component (implant, pillar, fixation screw of the pillar, cortical bone and trabecular bone) was modeled separately to allow the independent visualization and to verify the level of stress based on the color scale provided by the program. According to procedure adopted on the simulations by finite elements, it was considered the following simplifying hypothesis:

Table 1 - Differences of systems used on simulation.

Implant	Ar-Torq	Flash
Screw	Double	Double
Screw profile	Triangular	Trapezoidal
Pitch of screw	0.6	0.8
Apex	Conical	Conical
Body	Cylindrical	Conical
Coronary	Coronary	Micro partial screw
Mills ammount	4	2
Mills profile	Parallel	Helical

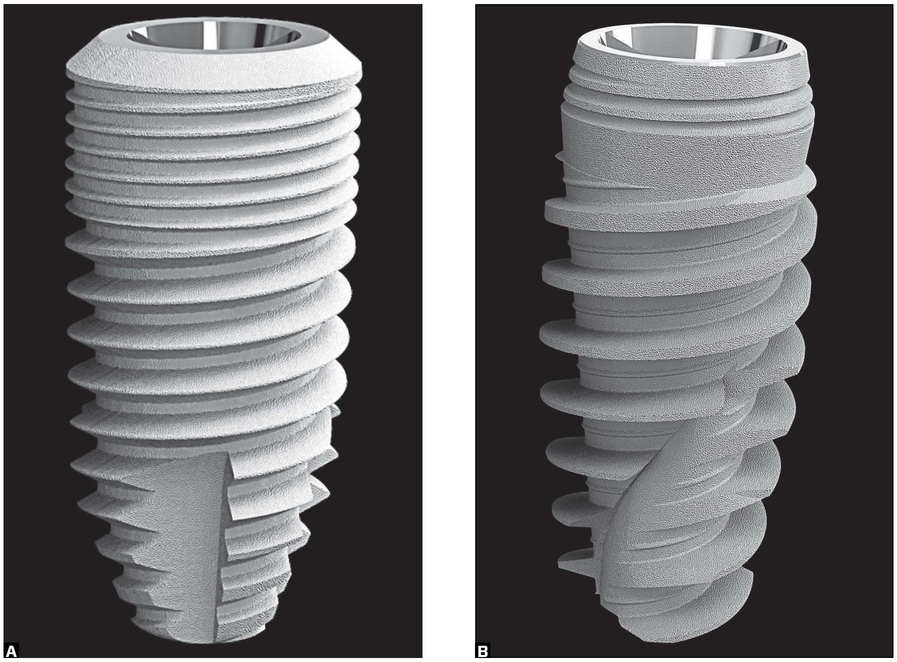

Figure 1 - Implant with NP platform used in simulation: **A**) Ar-Torq and **B**) Flash.

The materials were considered homogeneous, isotropic and linearly elastic. It was considered the implants manufactured in titanium commercially pure ASTM, degree 4 (ASTM F67); the pillar in titanium commercially pure ASTM, degree 2 (ASTM F67); and the fixation screw of the pillar in titanium, degree 5 (ASTM F136). The properties of the cortical and trabecular bones were the available in literature[10-13], which are displayed on Table 2. For analysis of results, the company Conexão Sistemas de Prótese released the technical information of the titanium provider on different categories of purity.

Table 2 - Mechanical properties of materials used.

Material	Modulus of elasticity (GPa)	Poisson's ratio
Titanium	110.00	0.35
Cortical bone	13.70	0.30
Trabecular bone	1.37	0.30

For the creation of the mesh, it was used the automatic system of the program with some changes on pre-programmed definitions by Ansys. The growth factor of the element was altered to prevent distortions on numeric results, besides proceeding to a local refining on the region of the interface implant-bone, considered a critical point. The finite element used was tetrahedral type. The contacts were defined as "bonded", i.e., it does not allow relative motion between pieces. The purpose of this choice was to determine a rigid union between volumes of the model. Regarding the number of elements and knots of the models, there was respectively 32988 and 63317 for the Ar-Torq system and 33671 and 61220 for the Flash system. According to data in literature,[1,2,3,10,14-19] the loadings used in the present work were of 100N[1,2,3,10,15-19] applied in three directions: axial, inclined to 15°[15,16] and 30°[20] in relation to longitudinal axis of the implant.

Besides the loading on the system, it was also applied the torque of 30N/cm of threading of the fixation screw of the pillar (pre-load), following the protocol suggested by the company. The simulation was multistep, i.e., first it was applied the torque on the screw and then it was applied the loading on the system. The constraint to motion was implemented on external areas of the cortical bone (Fig 2, in blue) to allow liberty of the system (implant and component). From the definitions, it was possible to use the program to calculate Von Mises equivalent stress on bones and on components of the implants systems.

Results

For analysis of results, it was used Von Mises equivalent stress. The stress values calculated on simulations are shown on Table 3. It was determined the maximum stress transmitted to the bone as well as maximum stress on implants and components (pillar and fixation screw of the pillar).

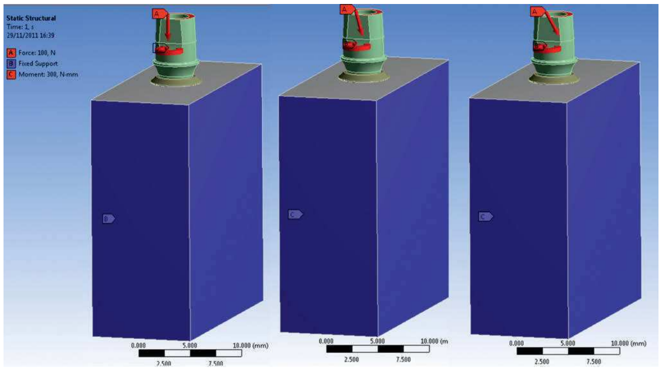

Figure 2 - Loads and restriction of simulated models.

On figures 3 to 5, it will be shown only images with stress distribution to the implant, cortical bone and trabecular bone on three different directions of loading. On the top of the images, it is found the results for the Ar-Torq system and on the bottom the results for the Flash. All figures were organized so that on the first column are the results for the axial loading, on the second column for the inclined to 15° loading and on the third column for the inclined to 30° loading.

To analyze the possibility of fracture of the implants and components, it was proposed to create a safety factor (SF) for different conditions of analysis. The SF is calculated through the relation between the value of stress in each point (σ_L) divided by the outflow limit (σ_e) of the material used on manufacturing. Using these criteria it was created figures that show values of the relation local stress and outflow limit ($SF = \sigma_e / \sigma_L$). It is shown on Figure 6 the values of SF calculated for different regions of the implant with loading inclined to 30°, this is the loading that presented the most critical condition.

Table 3 - Values of von Mises maximum stress (MPa) calculated on simulations of loading of several components of the models used.

	Direction of the applied force					
	Axial		Inclined 15°		Inclined 30°	
Platform	Ar-Torq	Flash	Ar-Torq	Flash	Ar-Torq	Flash
Implant	172.4	143.41	196.1	146.84	226.9	169.71
Pillar screw	350.2	350.3	350.7	354.62	351.2	358.54
Pillar	115.6	230.18	119.6	274.22	144.7	316.19
Cortical bone	15.3	13.79	17.9	16.49	23.6	20.16
Trabecular bone	2.2	1.48	2.7	1.56	3.2	1.85

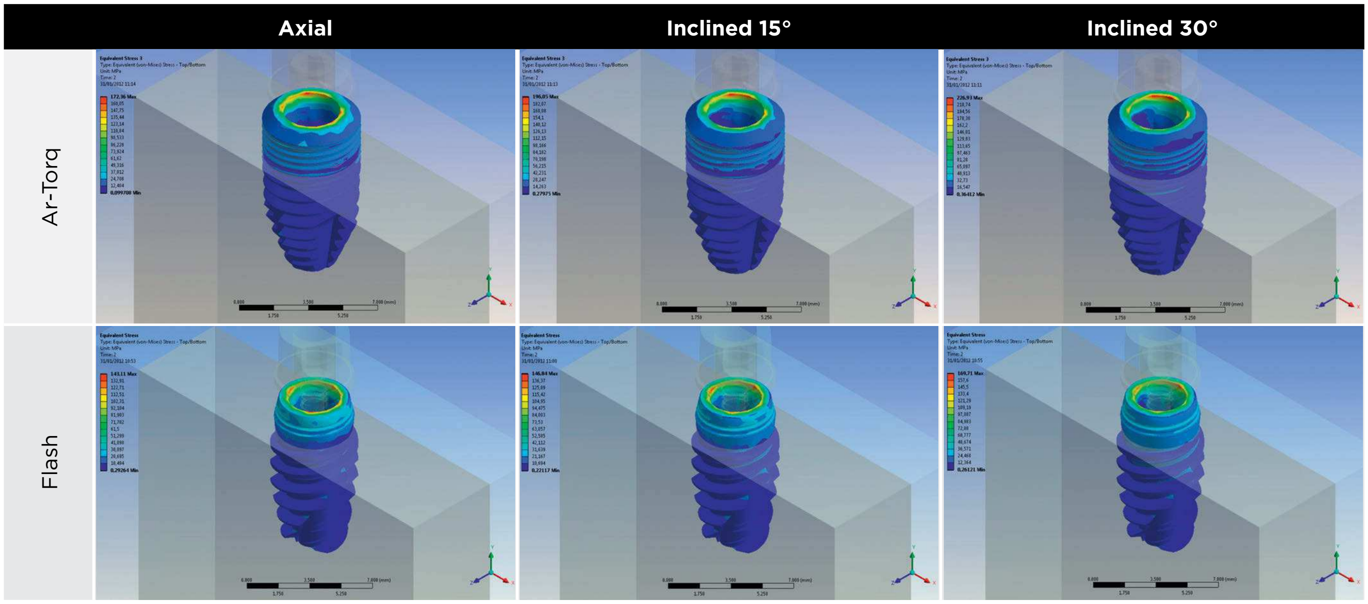

Figure 3 - Von Mises stress on implant with platform Ar-Torq and Flash on three directions.

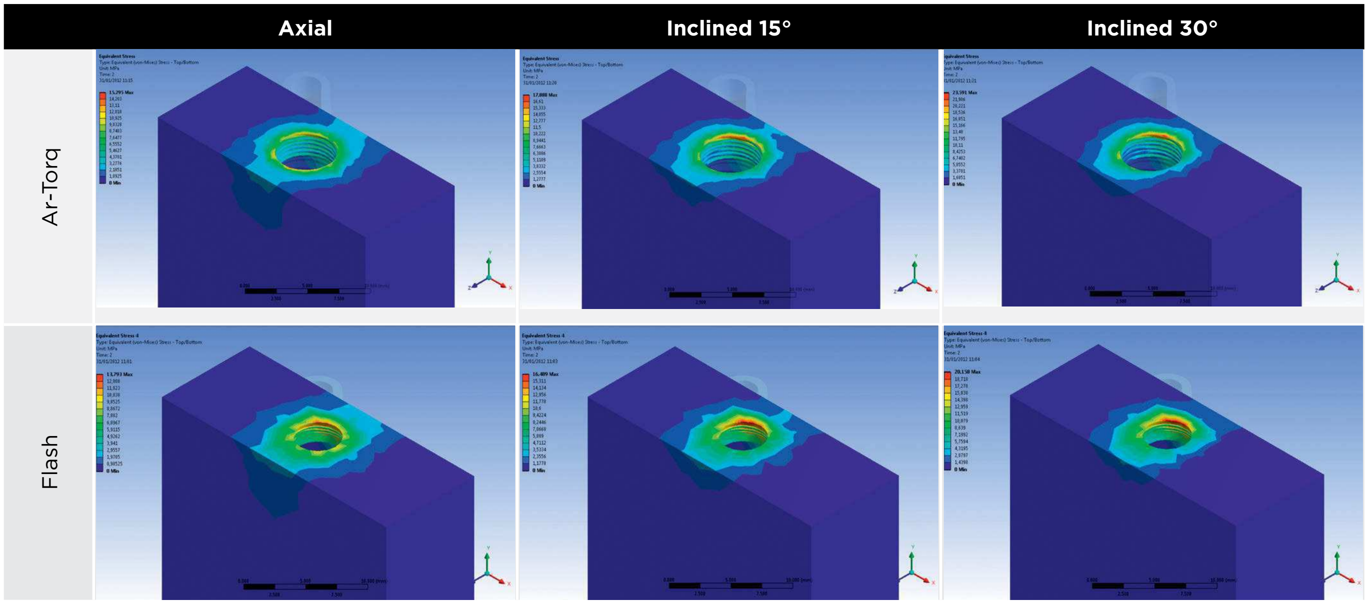

Figure 4 - Von Mises stress on cortical bone on the system with implant Ar-Torq and Flash on three directions.

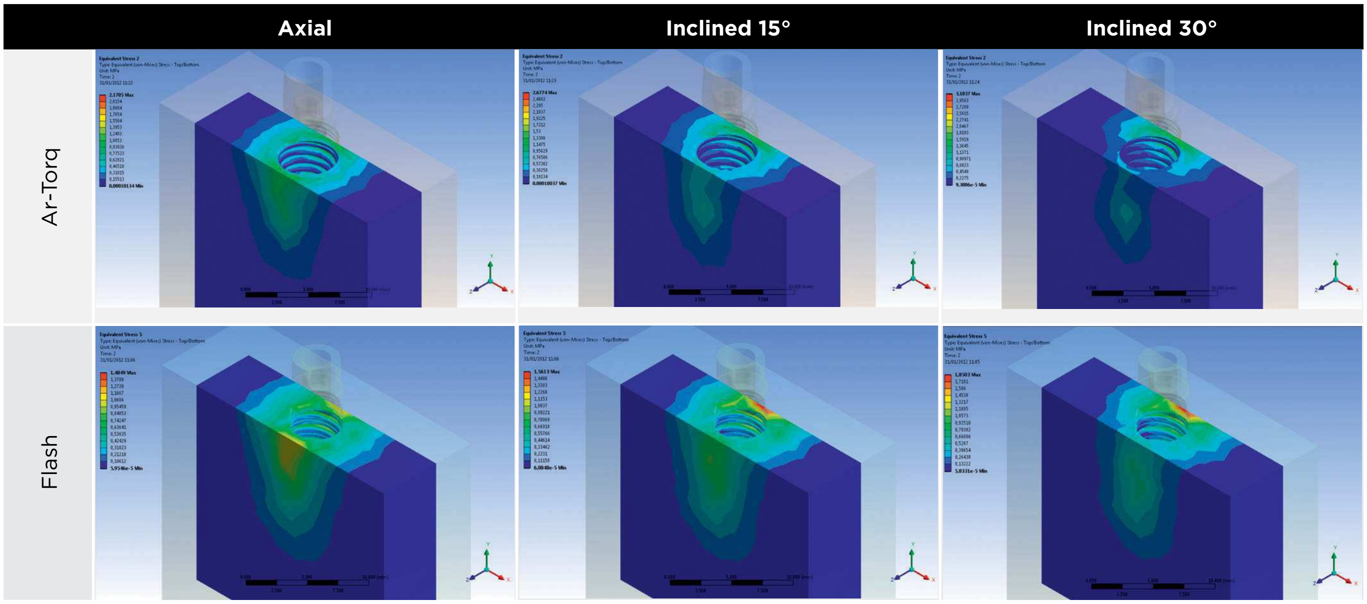

Figure 5 - Von Mises stress on trabecular bone on the system with implant Ar-Torq and Flash on three directions.

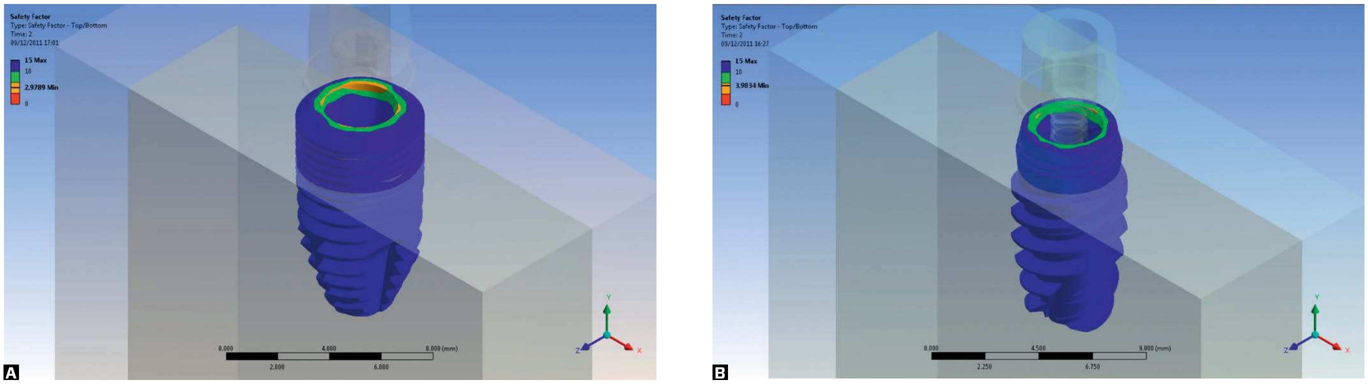

Figure 6 - Safety factor for implants **(A)** Ar-Torq and; **(B)** Flash.

Discussion

Considering that the dimension of the implants with internal prosthetic connections are more critical than conventional implants with external hexagon, on the present work the simulations were performed with implants with trade name Ar-Torq and Flash that contain internal hexagon. The intention to verify the influence of external geometric variation of dental implant is proven justified in previous work where it was used an asymmetric two-dimensional model to study the influence of the screw thread shape in relation to stress levels. It was found significant differences showing that the geometric variation of the model does influence on the distribution and on stress levels, proving that the ideal is to use implants with threads with slight curvature and lower depth.[21] The study of the differences of geometry and loadings applied was also explored in another work[22] that compared five systems of commercial implants. It was concluded that the error occurs on the marginal region of the implant, on the cortical bone and in compression. From the loading point of view, on literature it is possible to find experimental values of the bite force determined with electric extensometer.[14] It is known that this force increases during period of adaptation of the prosthesis. For computational simulation, most works presented in literature consider axial loadings[1,2,3,10,15-19] of 100N but some works consider the inclination of the equivalent loading to 15°[15,16] and 30°.[20] Literature presents study with simulation of finite elements where the model was submitted to a variation on the cusp inclination.[23] It was concluded that it is better to have a lower inclination of the cusp to obtain lower stress on the system bone-implant due to reduction of the force lateral components. This way, the choice of loading used in this work is relevant in relation to its intensity and direction. Through the use of simulations it is possible to observe the mechanical behavior of the prosthesis and estimate the success of the components before being commercialized. It is always sought to obtain values of stress lower than the material outflow limit to guarantee that there will not be plastic deformation of the components. Literature shows that the bone must be submitted to maximum stress of 167Mpa, this is the limit value to initiate the bone reabsorption.[24] This way it is possible to evaluate quantitatively the error possibility and qualitatively through figures with color gradient, the location of error occurrence of the implants systems. Regarding the evaluated prosthetic systems in the present work, it was not observed values superior to the limit of material outflow, showing that there will not be plastic deformation or fracture of the implants or any other component for loads of 100N on three implemented directions. It was also evidenced on figures 3,4 and 5 that the inclination of equivalent loading has influence on the stress distribution, as well as on its intensity. It was possible to observe on figures 4 and 5 that the values of the stress transmitted to the bone varies according to the loading used and to the type of dental implant evaluated. As the loading becomes more inclined, i.e., the components of the lateral forces increase, the stress on the bone and on the system also increase. Besides the method of finite elements, study involving photoelasticity was implemented to evaluate implants with different geometries.[25,26] It was verified that for loading with components of lateral forces, the stress are greater than for vertical loadings. This is due to reduction of bending moment created. In relation to the two types of dental implant evaluated, the Flash system transmits lower stress to the cortical bone, trabecular bone and to the dental implant. The fixation screw of the pillar had very similar behavior. For the pillar the behavior was the opposite, i.e., there was a greater effort on this component for the system with the implant Flash, however, in imposed conditions for this model, this effort does not influence the loss of component once the stresses were inferior to the limit of this material outflow. For analysis of safety factor, it was observed that both systems have high safety factor and would need a loading of 3 or 4 times more intense to begin to deform or components of laterality greater than 30° to intensificate the shear stress. According to what is shown on figure 6, for the loads used in this work, both systems presented high mechanical efficiency and the stress distribution was very similar for loadings on the same direction. For all simulations performed, the levels of stress show that there will not be bone resorption due to overload for being below the limit indicated on literature.[24]

Conclusion

Based on the performed computational simulations, it can be concluded that:

1. The system with dental implant Flash transmit lower stress to the bone and dental implant.
2. The safety factor of the implants is high suggesting it supports loadings more aggressive in intensity and directions.
3. As the loading becomes more inclined, i.e., the components of the lateral forces increase, the stress on the bone and on the prosthetic components increase.
4. For all simulations performed, the systems behaved appropriately so there is no indication of deformation or fracture on the prosthetic components or even bone resorption due to overload.

REFERENCES

1. Lehmann RB, Elias CN. Uso de simulação 3D no desenvolvimento de implantes e sistemas protéticos com pilares retos e angulados. ImplantNews. 2010;7(1):113-20.
2. Lehmann RB, Elias CN, Teixeira MF. Efeito da oclusão cêntrica em um sistema rígido de esplintagem dos implantes para carga imediata pelo método de elementos finitos. Rev Bras Implant. 2009;15(1):6-10.
3. Lehmann RB, Elias CN. Tensões em implantes cônicos com hexágono externo e com hexágono interno. Rev Dental Press Periodontia Implantol. 2008;2(2):91-9.
4. Hansson S, Werke M. The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study. J Biomech. 2003;36(9):1247-58.
5. Alkan I, Sertgo A, Ekici B. Influence of occlusal forces on stress distribution in preloaded dental implant screws. J Prosthet Dent. 2004;91(4):319-25.
6. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress / strain distribution in bone around implants: A 3-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2003;18(3):357-68.
7. Sahin S, Cehreli MC, Yalçin E. The influence of fuctional forces on the biomechanics of implant-supported prostheses - a review. J Dent. 2002;30(7-8):271-82.
8. Stanford CM, Brand RA. Toward an understanding of implant occlusion and strain adaptive bone modeling and remodeling. J Prosthet Dent. 1999;81(5):553-61.
9. Vaillancourt H, Pillar RM, McCammond D. Factors affecting crystal bone loss with dental implants partially covered with a porous coating: a finite element analysis. Int J Oral Maxillofac Implants. 1996;11:351-9.
10. Lehmann RB, Elias CN. Simulação da distribuição de tensões em implante dentário curto (5,7mm x 5,0mm) com carregamento axial em corticais com diferentes espessuras. Dental Press Implantology. 2011;5(4):46-54.
11. Caglar A, Bal BT, Aydin C, Yilmaz H, Ozkan S. Evaluation of stresses occurring on three different zirconia dental implants — three-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2010;25(1):95-103.
12. Cruz M, Wassall T, Toledo EM, da Silva Barra LP, Cruz S. Finite element stress analysis of dental prostheses supported by straight and angled implants. Int J Oral Maxillofac Implants. 2009;24(3):391-403.

13. Silva GC, Mendonça JA, Lopes LR, Landre J Jr. Stress patterns on implants in prostheses supported by four or six implants: a three-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2010;25(2):239-46.
14. Iplikçioglu H, Alça K. Comparative evaluation of the effect of diameter, length and number of implants supporting three-unit fixed partial prostheses on stress distribution in the bone. J Dent. 2002;30:41-6.
15. Schrotenboer J, Tsao YP, Kinariwala V, Wang HL. Effect of platform switching on implant crest bone stress — a finite element analysis. Implant Dent. 2009;18(3):260-9.
16. Lee CC, Lin SC, Kang MJ, Wu SW, Fu PY. Effects of implant threads on the contact area and stress distribution of marginal bone. J Dent Sci. 2010;5(3):156-65.
17. Li T, Kong L, Wang Y, Hu K, Song L, Liu B, et al. Selection of optimal dental implant diameter and length in type IV bone: a three dimensional finite element analysis. Int J Oral Maxillofac Surg. 2009;38(10):1077-83. Epub 2009 Aug 4.
18. Bujtár P, Sándor GK, Bojtos A, Szucs A, Barabás J. Finite element analysis of the human mandible at 3 different stages of life. Oral Surg Oral Med Oral Pathol Oral Radiol Endod. 2010;110(3):301-9. Epub 2010 May 1.
19. Ao J, Li T, Liu Y, Ding Y, Wu G, Hu K, et al. Optimal design of thread height and width on an immediately loaded cylinder implant: A finite element analysis. Comput Biol Med. 2010;40(8):681-6.
20. Eser A, Tonuk E, Akca K, Cehreli MC. Predicting time-dependent remodeling of bone around immediately loaded dental implants with different designs. Med Eng Phys. 2010;32(1):22-31. Epub 2009 Nov 1.
21. Hansson S, Werke M. The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study. J Biomech. 2003;36(9):1247-58.
22. 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 Dec;92(6):523-30.
23. Moraes MCCB, Moraes EJ, Elias CN. Análise de tensões em implantes osseointegrados por elementos finitos: variação da inclinação da cúspide e largura da mesa oclusal. Rev Bras Implant. 2002;8(2):21-6.
24. Ciftçi Y, Canay S. The effect of veneering materials on stress distribution in implant supported fixed prosthetic restorations. Int J Oral Maxillofac Implants. 2000;15(4):571-82.
25. Gross MD, Nissan J, Samuel R. Stress distribution around maxillary implants in anatomic photoelastic models of varying geometry Part I. J Prosthet Dent. 2001;85(5):442-9.
26. Gross MD, Nissan J. Stress distribution around maxillary implants in anatomic photoelastic models of varying geometry Part II. J Prosthet Dent. 2001;85(5):450-4.