Saudi Dental Journal (2022) **34**, 579–584

King Saud University

Saudi Dental Journal

www.ksu.edu.sa
www.sciencedirect.com

ORIGINAL ARTICLE

Stress analysis and factor of safety in three dental implant systems by finite element analysis

E. Menacho-Mendoza [a], **R. Cedamanos-Cuenca** [b], **A. Díaz-Suyo** [a,*]

[a] *Facultad de Ciencias de Vida y Salud, Universidad Científica del Sur, Lima, Perú*
[b] *Facultad de Ingeniería Mecánica, Universidad Nacional de Ingeniería, Lima, Perú*

Received 8 June 2021; revised 19 August 2022; accepted 21 August 2022
Available online 27 August 2022

KEYWORDS

Factor of safety;
Stress distribution;
Finite element analysis;
von Mises stress;
Dental implants

Abstract *Objective:* The purpose of this study was to compare the stress distribution and the factor of safety of three dental implant systems using the finite element method.

Materials and methods: Three commercial dental implant systems were designed using Solid Works 2020 software: Model A with an internal octagonal connection and matching platform, Model B with an internal hexagon connection and switching platform, and Model C with an internal 15° conical-cylindrical connection and switching platform. A 200 N load was applied to each design in both axial and 30° oblique directions using the finite element method.

Results: In the three dental implant systems, the maximum von Mises stress was concentrated at the cervical level of the bone-implant interface in all models. Model C showed lower maximum stress values in both axial and 30° oblique loads. The highest maximum stress value was observed with the application of the oblique load in all the study models, and the factor of safety was less than one in Model A when subjected to a 200 N oblique load.

Conclusion: The switching platform models generated lower maximum stress values and a factor of safety higher than one which is considered an acceptable value.

Clinical relevance: A dental implant system with an internal hexagon or conical connection and a switching platform generates lower maximum von Mises stress values both on the implant components and on the *peri*-implant tissues.

© 2022 The Authors. Production and hosting by Elsevier B.V. on behalf of King Saud University. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

1. Introduction

The finite element method (FEM) has been widely used to evaluate the biomechanical behavior of orthodontic appliances, fixed prostheses, total dentures, and dental implants as well as to analyze stress distribution in *peri*-implant bone and natural or restored teeth (Knop et al., 2015; Matsuoka et al., 2021; Reddy et al., 2019; Romanyk et al., 2020). Generally, the FEM concentrates on the stress distribution, which is related to stress variations along the structure of a solid. In

* Corresponding author at: Jr Guillermo Biela 2044, San Martin de Porres, Lima, Perú.
E-mail address: jdiazsu@cientifica.edu.pe (A. Díaz-Suyo).

Peer review under responsibility of King Saud University. Production and hosting by Elsevier.

Production and hosting by Elsevier

https://doi.org/10.1016/j.sdentj.2022.08.006
1013-9052 © 2022 The Authors. Production and hosting by Elsevier B.V. on behalf of King Saud University.
This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

addition, stress acts as a measure to know the intensity of force that an object supports (Franco-Silva and Ángeles-Maslucán, 2014). Another variable that can be evaluated using the FEM is the factor of safety (FOS), this is the ratio between the yield stress of the material and the maximum von Mises stress (Herza et al., 2018; Darwich et al., 2021).

Most studies have evaluated the maximum stress using chewing forces in normal ranges with different dental implant dimensions and prosthetic structures (Cho et al., 2016; Farronato et al., 2019; Kharsan et al., 2019; Pournasrollah et al., 2019; Fiorillo et al., 2020; Anitua et al., 2021). A previous study determined that the implant-abutment connection is a key point in long-term clinical success (Kharsan et al., 2019; Pournasrollah et al., 2019), whereas others studies reported that stress decreases as the implant diameter increases (Cho et al., 2016; Anitua et al., 2021). Another investigation developed a novel conical dental implant connection that generated lower maximum stress values both on the implant components and on the *peri*-implant tissues (Fiorillo et al., 2020). Only one study reported the FOS in matching and switching platform implants; it was established that the most critical area was concentrated on the implant neck in matching implants (Farronato et al., 2019). The purpose of this study was to compare the stress distribution and FOS of three dental implant systems (abutment, screw, and dental implant) using FEM.

2. Materials and methods

The study was carried out using FEM and three study models were built. The study consisted of two experimental groups (Models A and B) and a control group (Model C). The parameters used for the construction of the models were as follows:

Model A (Sweden & Martina Global): 4.3 mm × 10 mm implant with internal octagonal connection and matching platform, and a preformed abutment with screw.

Model B (Dentium SuperLine): 4 mm × 10 mm implant with internal hexagon connection and switching platform, and a milling wearable abutment with screw.

Model C (Straumann): 4.1 mm × 10 mm BLT implant with internal 15° conical-cylindrical connection and switching platform, and a cementable abutment with screw.

A type IV bone block was designed with a diameter of 20 mm and a height of 13 mm, and the thickness of the cortical bone used was 2 mm. Each dental implant was integrated into the bone block to perform the finite element method.

To replicate the geometry of the components of each dental implant system (abutment, screw, and dental implant), preexisting plans were requested from the implant manufacturing companies. Step files and design maps were used in Models A and B, respectively. Model C was designed based on a physical implant using measuring instruments, such as Vernier, micrometer, magnifying glasses, and thread tester. This procedure was executed by the trained professional. The designed components were assembled and meshed by a series of Jacobian points using finite element software (SolidWorks 2020). A 3D solid-type mesh with tetrahedral elements based on curvature was used, which provided greater precision in the analysis of results due to its automatic creation of more elements in areas of greater curvature adapted to the circumferential shapes of the implants. The number of nodes and elements are listed in Table 1. An axial load of 200 N and an oblique load of 200 N at an angle of 30° were applied to the surface of the abutment based on the literature (Cho et al., 2016; Aguilar et al., 2019; Aslam et al., 2019) (Fig. 1). The Young's modulus, Possion's rate and yield stress were selected based on the literature (Pellizzer et al., 2012; Pellizzer et al., 2013) and are shown in Table 2.

The stress distribution was assessed using the von Mises stress through the comparison of normal, principal, and equivalent stresses. The FOS was calculated as the ratio between the yield stress of the material and the maximum von Mises stress. A FOS greater than one means that the material deforms only elastically which is a main requirement to prevent immediate mechanical damage.

3. Results

3.1. Stress analysis

An axial and oblique loads were applied on the surface of the abutment to analyze the stress distribution and determine the maximum stress values; the stress distribution was evaluated using von Mises stress. After applying an axial load, Model A showed a maximum von Mises stress value of 282.2 MPa (Table 3), and the critical stress concentration was located in the transmucosal portion of the abutment (Fig. 2). Models B and C showed maximum von Mises stress values of 183.2 MPa and 216.7 MPa, respectively (Table 3); this stress was concentrated on the implant neck in both models (Fig. 2).

After applying an oblique load, Model A showed a maximum von Mises stress value of 1312 MPa (Table 3), this stress was located in the transmucosal portion of the abutment (Fig. 2). Models B and C showed maximum von Mises stress values of 573.7 MPa and 373.4 MPa, respectively (Table 3);

Table 2 Materials and mechanical properties.

Material	Young's Modulus (MPa)	Possion's Rate	Yield Stress (MPa)
Titanium Grade V	110.000	0.35	870
Cortical Bone	13.700	0.3	150
Cancellous Bone	1.370	0.3	130

Table 1 Number of nodes and elements of each study model.

MODEL A		MODEL B		MODEL C	
Nodes	Elements	Nodes	Elements	Nodes	Elements
169,764	106,881	142,232	89,402	60,714	36,691

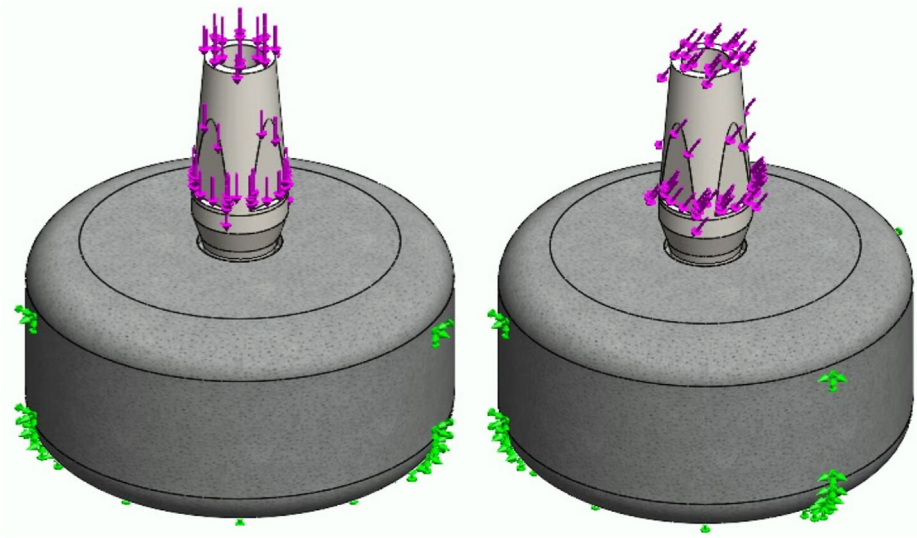


Fig. 1 Simulation of axial and oblique (30°) load application in Models A and B.

the stress was concentrated on the implant neck in both models (Fig. 2). Moreover, the switching platform models showed lower maximum von Mises stress values than the matching platform model when subjected to both loads.

3.2. Factor of safety

After applying an axial load, Models A, B, and C showed FOS values of 2, 2.6, and 2.7, respectively (Table 3); the most critical areas were concentrated on the implant-abutment interface and the surrounding cortical bone (Fig. 3). After applying an oblique load, Models B and C showed FOS values of 1.1 and 1.4, respectively (Table 3); the most critical zones were located in the transmucosal portion of the abutment, implant neck and the surrounding cortical bone (Fig. 3). Only Model A showed a FOS value lower than one, the most critical area was located in implant neck and the surrounding cortical bone (Fig. 3). The switching platform models showed higher FOS values than the matching platform model.

4. Discussion

In oral implantology, FEM is used with the aim of preventing failures and complications in dental implants, implant-supported fixed dental prostheses, and implant-supported overdentures. Furthermore, FEM allows the measurement of the stress distribution in various dental implant designs and bone during chewing (Chang et al., 2018).

In general, the models with a switching platform (Model B and C) showed lower maximum stress values in both axial and oblique loads in the implant at the implant-abutment interface level. On the other hand, the abutment of Model A showed higher stress under both loads. This finding is related to the design of the implant system itself, which prevents the screw from receiving all the stress, causing the abutment to receive all the load.

Regarding the bone, the maximum von Mises stress was located in the cortical bone surrounding the cervical area of the implant in all three study models. The results of the present study are consistent with a previous report that found that the highest stress was concentrated in the cortical bone next to the implant neck (Yamanishi et al., 2014). Another study found that the dissipation of stress in cortical bone is limited to the immediate area that surrounds the implant due to its greater resistance to deformation with respect to trabecular bone (Danza et al., 2009). In all three models, the maximum stress value in the bone was lower than the yield stress of the cortical bone, except for the matching platform implant subjected to an oblique load. If the maximum stress value in the bone is

Tabla 3 Maximum von Mises stress and FOS in all three study models.

		MAXIMUM VON MISES STRESS					FOS		
		Study model	Bone	Implant	Abutment	Screw	Study model	Implant	Abutment and Screw
MODEL A	Axial	282.3	75.4	160.7	282.3	0	2	5.4	3.1
	Oblique	1312	287.7	605.4	1312	0	0.52	1.4	0.66
MODEL B	Axial	183.2	58.4	183.2	64.8	38.4	2.6	4.7	10
	Oblique	573.7	141.8	573.7	312.8	126.3	1.1	1.5	2.8
MODEL C	Axial	216.7	55.5	216.7	41.9	28.4	2.7	4	10
	Oblique	373.4	107	373.4	188.6	52.3	1.4	2.3	4.6

Note: Maximum von Mises stress in MPa.

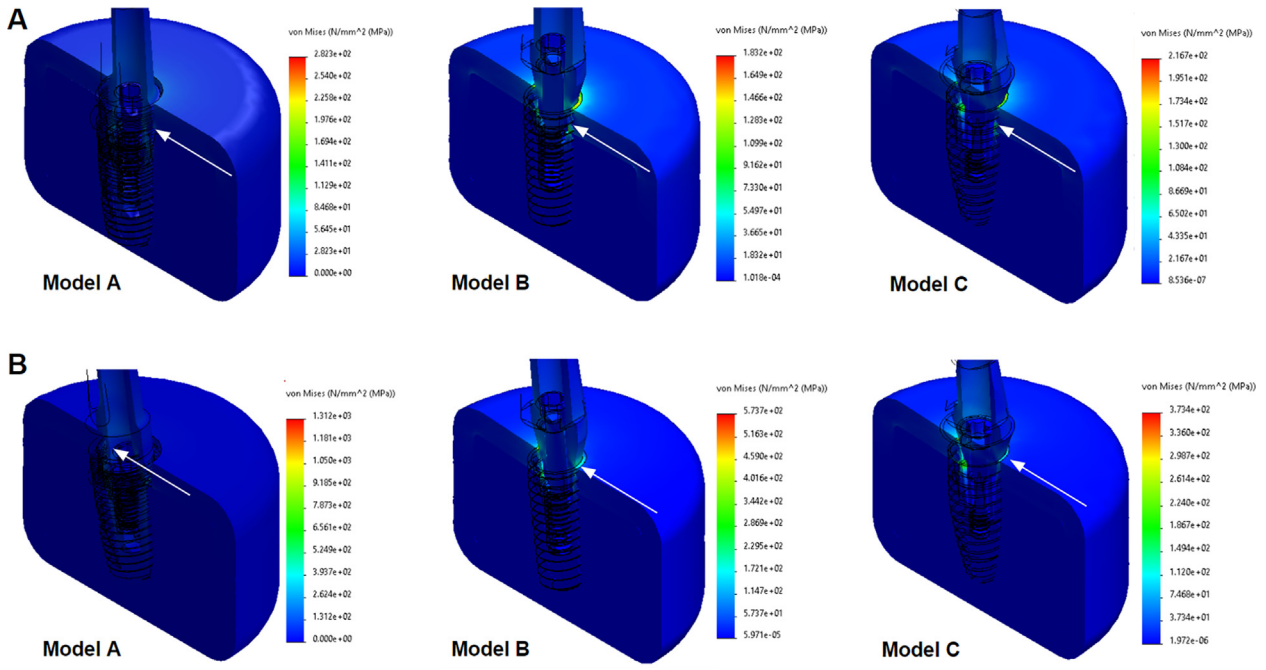


Fig. 2 Stress map, the arrows show areas of major stress. A) Models A, B, and C subjected to a 200 N axial load. B) Models A, B, and C subjected to a 200 N oblique load.

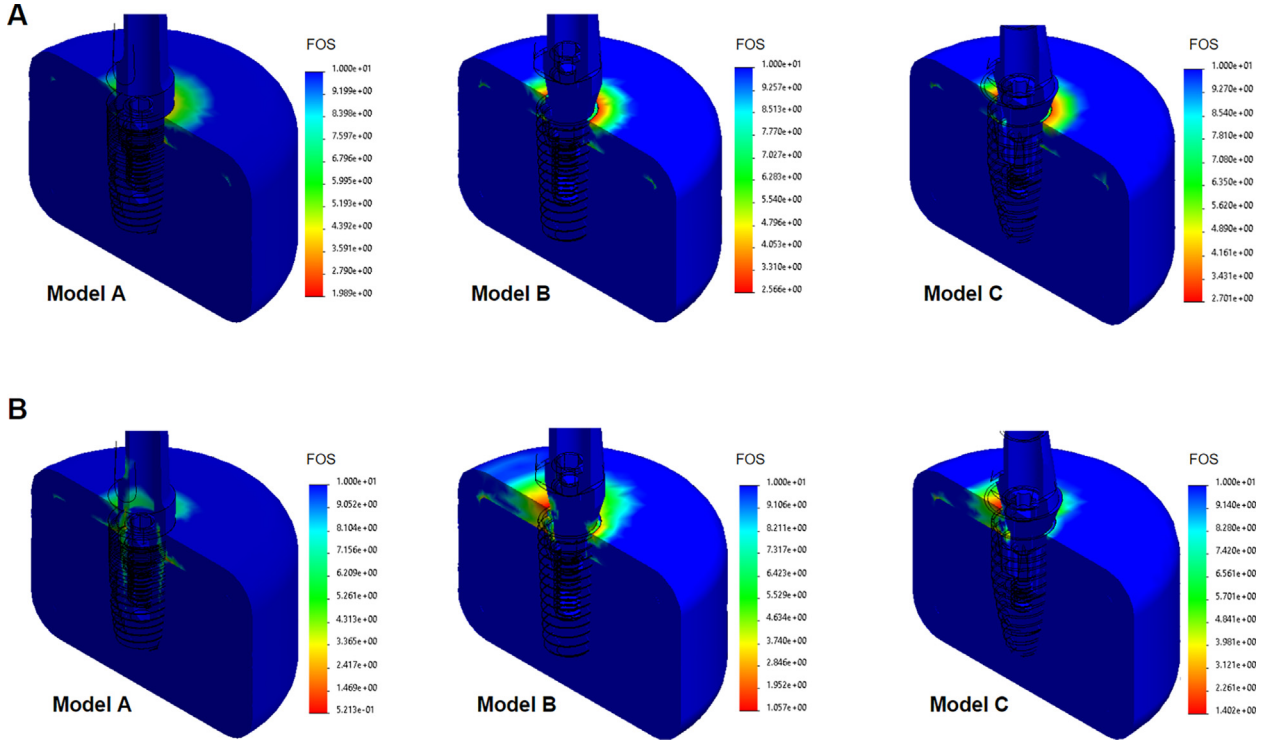


Fig. 3 FOS map, the most critical zones are shown in red A) Models A, B, and C subjected to a 200 N axial load B) Models A, B, and C subjected to a 200 N oblique load.

higher than the yield stress of the cortical bone, this maximum stress would generate a plastic deformation in the bone which could lead to a marginal bone resorption.

Previous studies have shown that the presence of the switching platform reduces stress in the bone. The results of the present study showed lower von Mises stress values in Models B and C compared to Model A, which presented a matching platform. A previous study determined that switching platform reduces the maximum stress level on the cortical bone, this might be a biomechanical explanation of the minimization of the *peri*-implant bone resorption (Bouazza et al., 2015). Another study found that switching platform decreased the stress within the *peri*-implant bone and may decrease marginal bone resorption (Aslam et al., 2019).

The implant-abutment connection plays a crucial role in stress distribution. The reduction of stress at the implant-abutment connection may avoid some mechanical complications, such as abutment fracture, screw fracture, screw loosening, and augmented leakage at the implant-abutment connection (Kharsan et al., 2019). A study found that implants with conical connections showed lower stress than implants with internal hexagonal connections. The internal conical connection generated greater resistance to deformation and fracture when subjected to oblique loads. However, the internal hexagonal connection is considered a suitable option (Coppedê et al., 2009; Schmitt et al., 2014). Another study found that the tapered connection showed a more homogeneous pattern (Farronato et al., 2019). The flat-to-flat implant-abutment connection is mechanically less stable than an internal conical implant–abutment connection and generates the worst sealing at the interface (Coppedê et al., 2009).

Regarding FOS, Model A presented a FOS less than one only when subjected to an oblique load of 200 N. The most critical component was the abutment which had a FOS value less than one and a maximum stress value higher than the yield stress. Stress concentration would generate a plastic deformation or failure of the abutment in Model A which is matching platform. On the other hand, the switching platform implants had higher FOS values than the matching platform implant. According to the results, the critical zone was located next to the implant-abutment connection in all three models that had a FOS value higher than one, except for Model A subjected to an oblique load. These results are in agreement with a study that determined that the most critical zones were present in the most coronal part of the implant-abutment interface and in the implant neck (Farronato et al., 2019).

Among the limitations of the study, the conditions of the stomatognathic system could not be imitated in a complete way because other possible factors, such as horizontal or oblique loads with different angulations or the presence of masticatory muscles, were not taken into account.

5. Conclusion

Models with switching platforms with an internal hexagon or conical-cylindrical connection generate lower maximum stress values, the major areas of stress were concentrated on the implant-abutment interface and the surrounding cortical bone. Furthermore, switching platform models showed FOS values higher than one which do not generate plastic deformation of the components of the dental implant system.

Funding

This research was self-financed.

CRediT authorship contribution statement

E. Menacho-Mendoza: Conceptualization, Data curation, Formal analysis, Funding acquisition, Investigation, Methodology, Project administration, Resources, Software, Validation, Visualization, Writing – original draft, Writing – review & editing. **R. Cedamanos-Cuenca:** Conceptualization, Data curation, Formal analysis, Investigation, Methodology, Project administration, Software, Supervision, Validation, Visualization, Writing – original draft, Writing – review & editing. **A. Díaz-Suyo:** Conceptualization, Data curation, Formal analysis, Investigation, Methodology, Project administration, Software, Supervision, Validation, Visualization, Writing – original draft, Writing – review & editing.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

Aguilar, J.I., Villarraga, J.A., Latorre, F., 2019. Evaluación de la microdeformación de la zona maxilar anterior con regeneración. Análisis de elementos finitos. Odontol. Sanmarquina. 22 (2), 110–116. https://doi.org/10.15381/os.v22i2.16223.

Anitua, E., Larrazabal, N., Morales, I., Saracho, L., 2021. Influence of dental implant diameter and bone quality on the biomechanics of single-crown restoration. A Finite Element Analysis. Dent. J. 9 (9), 103–113. https://doi.org/10.3390/dj9090103.

Aslam, A., Hassan, S.H., Aslam, H.M., Khan, D.A., 2019. Effect of platform switching on peri-implant bone: A 3D finite element analysis. J. Prosthet. Dent. 121 (6), 935–940. https://doi.org/10.4317/jced.52539.

Bouazza, K., Martínez, G., Peiró, G., Ródenas, J., 2015. Effect of platform switching on the peri-implant bone: A finite element study. J. Clin. Exp. Dent. 7 (4), 483–488. https://doi.org/10.4317/jced.52539.

Chang, Y., Tambe, A.A., Maeda, Y., Wada, M., Gonda, T., 2018. Finite element analysis of dental implants with validation: to what extent can we expect the model to predict biological phenomena? A literature review and proposal for classification of a validation process. Int. J. Implant. Dent. 4 (1), 1–14. https://doi.org/10.1186/s40729-018-0119-5.

Cho, S.Y., Huh, Y.H., Park, C.J., Cho, L.R., 2016. Three-Dimensional Finite Element Analysis of the Stress Distribution at the Internal Implant-Abutment Connection. Int. J. Periodontics. Restorative. Dent. 36 (3), 49–58.

Coppedê, A.R., Bersani, E., De Mattos, M.G., Rodrigues, R.C., Sartori, I.A., Ribeiro, R.F., 2009. Fracture resistance of the implant–abutment connection in implants with internal hex and internal conical connections under oblique compressive loading: An in vitro study. Int. J. Prosthodont. 22 (3), 283–286.

Danza, M., Palmieri, A., Farinella, F., Brunelli, G., Carinci, F., Girardi, A., Spinelli, G., 2009. Three dimensional finite element analysis to detect stress distribution in spiral implants and surrounding bone. Dent. Res. J. (Isfahan) 6 (2), 59–64.

Darwich, A., Alammar, A., Heshmeh, O., Szabolcs, S., Nazha, H., 2021. Fatigue Loading Effect in Custom-Made All-on-4 Implants

System: A 3D Finite Elements Analysis. IRBM. 42 (3), 1–8. https://doi.org/10.1016/j.irbm.2021.06.008.

Farronato, D., Manfredini, M., Stevanello, A., Campana, V., Azii, L., Farronato, M., 2019. A Comparative 3D Finite Element Computational Study of Three Connections. Materials (Basel). 12 (19), 31–35. https://doi.org/10.3390/ma12193135.

Fiorillo, L., Cicciù, M., D'Amico, C., Mauceri, R., Oteri, G., Cervino, G., 2020. Finite element method and von Mises investigation on bone response to dynamic stress with a novel conical dental implant connection. BioMed. Res. Int. 1, 1–13. https://doi.org/10.1155/2020/297606.

Franco-Silva, B.M., Ángeles-Maslucán, R.J., 2014. Análisis de tensiones con el método de elemento finito a nivel de la interfase hueso - implante de un nuevo diseño de conexión interna: Octodecágono. Rev. Cient. Odontol. (Lima) 2 (1), 127–135. https://doi.org/10.21142/2523-2754-0201-2014-%25p.

Herza, J., Ashley, M., Thorp, J., 2018. Factor of Safety? - Do we use it correctly? URL: https://www.researchgate.net/publication/327920961_Factor_of_Safety-Do_we_use_it_correctly.

Kharsan, V., Bandgar, V., Mirza, A., Jagtiani, K., Dhariwal, N., Kore, R., 2019. Comparative Evaluation of Three Abutment-Implant Interfaces on Stress Distribution in and Around Different Implant Systems: A Finite Element Analysis. Contemp. Clin. Dent. 10 (4), 590–594. https://doi.org/10.4103/ccd.ccd_739_18.

Knop, L., Gandini, L.G., Shintcovsk, R.L., Gandini, M.R., 2015. Scientific use of the finite element method in Orthodontics. Dental. Press. J. Orthod. 20 (2), 119–125.

Matsuoka, T., Nakano, T., Yamaguchi, S., Ono, S., Watanabe, S., Sato, T., Yatani, H., 2021. Effects of implant-abutment connection type and inter-implant distance on inter-implant bone stress and microgap: three-dimensional finite element analysis. Materials. 14 (9), 2421–2431. https://doi.org/10.3390/ma14092421.

Pellizzer, E.P., Verri, F.R., Falcón-Antenucci, R.M., Júnior, J.F., De Carvalho, P.S., De Moraes, S.L., Noritomi, P.Y., 2012. Stress analysis in platform-switching implants: a 3-dimensional finite element study. J. Oral. Implantol. 38 (5), 587–594. https://doi.org/10.1563/AAID-JOI-D-10-00041.

Pellizzer, E.P., Verri, F.R., De Moraes, S.L., Falcón-Antenucci, R.M., De Carvalho, P.S., Noritomi, P.Y., 2013. Influence of the implant diameter with different sizes of hexagon: analysis by 3-dimensional finite element method. J. Oral. Implantol. 39 (4), 425–431. https://doi.org/10.1563/AAID-JOI-D-10-00103.

Pournasrollah, A., Negahdari, R., Gharekhani, V., Torab, A., Jannati, S., 2019. Investigating the effect of abutment-implant connection type on abutment screw loosening in a dental implant system using finite element methods. J. Dent. Res. Dent. Clin. Dent. Prospects. 13 (4), 289–297. https://doi.org/10.15171/joddd.2019.044.

Reddy, M.S., Sundram, R., Eid Abdemagyd, H.A., 2019. Application of finite element model in implant dentistry: A systematic review. J. Pharm. Bioallied. Sci. 11 (2), 85–91.

Romanyk, D.L., Vafaeian, B., Addison, O., Adeeb, S., 2020. The use of finite element analysis in dentistry and orthodontics: Critical points for model development and interpreting results. Semin Orthod. 26 (3), 162–173. https://doi.org/10.1053/j.sodo.2020.06.014.

Schmitt, C.M., Nogueira-Filho, G., Tenenbaum, H.C., Lai, J.Y., Brito, C., Doring, H., Nonhoff, J., 2014. Performance of conical abutment (Morse Taper) connection implants: A systematic review. J. Biomed. Mater. Res. A. 102 (2), 552–574. https://doi.org/10.1002/jbm.a.34709.

Yamanishi, Y., Yamaguchi, S., Imazato, S., Nakano, T., Yatani, H., 2014. Effects of the implant design on peri-implant bone stress and abutment micromovement: three-dimensional finite element analysis of original computer-aided design models. J. Periodontol. 85 (9), 333–338. https://doi.org/10.1902/jop.2014.140107.