Comparative mechanical behavior of grand Morse and external hexagon systems dental implants in posterior maxillary region. A Three-Dimensional Finite Element Analysis (3D-FEA) study

Comportamento mecânico comparativo de implantes dentários dos sistemas grand Morse e hexágono externo na região posterior do maxilar. Um estudo de análise de elementos finitos tridimensionais (3D-FEA)

Comportamiento mecánico comparativo de implantes dentales con sistemas grand Morse y hexágono externo en la región maxilar posterior. Un estudio de análisis tridimensional por elementos finitos (3D-FEA)

Received: 09/25/2024 | Revised: 10/07/2024 | Accepted: 10/09/2024 | Published: 10/13/2024

Nicole Casalle
ORCID: https://orcid.org/0000-0001-8866-4167
São Paulo State University, Brazil
E-mail: nicolecasalle@hotmail.com
Lucas David Galvani
ORCID: https://orcid.org/0000-0003-4735-8045
Estácio de Sá University Center, Brazil
E-mail: lucas.galvani@icloud.com
Mário Augusto Pires Vaz
ORCID: https://orcid.org/0000-0002-6347-9608
University of Porto, Portugal
Email: gmavaz@fe.up.pt
Leonardo Mendes Ribeiro Machado
ORCID: https://orcid.org/0000-0002-3531-9471
Renato Archer Information Technology Center, Brazil
E-mail: leonardomrmachado@gmail.com
Pedro Yoshito Noritomi
ORCID: https://orcid.org/0000-0001-7333-3445
State University of Campinas, Brazil
E-mail: pedro.noritomi@cti.gov.br
Luís Geraldo Vaz
ORCID: https://orcid.org/0000-0003-0916-1962
São Paulo State University, Brazil
E-mail: geraldo.vaz@unesp.br

Abstract
The increasing use of dental implants has boosted demand for innovative prosthetic solutions. These new forms and approaches aim to improve clinical handling and treatment quality, offering better results and greater efficiency in patient care. Among recent developments, the grand Morse connection has become popular, yet no scientific studies have compared its mechanical performance with external hexagon implants or Morse cone implants. The objective of this study was to comparatively assess the mechanical characteristics of two types of prosthetic connections (external hexagon and grand Morse) in posterior maxilla rehabilitation, employing the finite element method: external hexagon and grand Morse (4.0 mm x 8.0 mm - Hélix Neodent), both with screw-retained crowns. Virtual models were generated with CAD software Rhinoceros 7® based on a posterior maxillary segment’s bone shape from the BioCAD protocol. A primary-order tetrahedral 3D mesh was created for analysis, simulating 100N loads at a 30º angle to the implant axis. Results showed greater shifting in the external hexagon model (0.1399 mm) compared to the grand Morse model (0.0208 mm). Stress analysis revealed similar patterns near the implant platform, but the external hexagon model exhibited higher von Mises stress (148.4 MPa) compared to the grand Morse model (99.03 MPa), which had better stress dispersion. Maximum Principal Stress was higher for the grand Morse model. Thus, the implant platform connection design affects stress distribution and intensity.
Keywords: Finite element analysis; Dental prosthesis, implant-supported; Dental implant; Maxilla.

Resumo
O crescente uso de implantes dentários tem impulsionado a demanda por soluções protéticas inovadoras. Essas novas formas e abordagens visam aprimorar tanto o manuseio clínico quanto a qualidade do tratamento, oferecendo melhores resultados e maior eficiência no atendimento ao paciente. Entre os desenvolvimentos recentes, a conexão Morse grande se tornou popular, mas não há estudos científicos que tenham comparado seu desempenho mecânico com o dos implantes de hexágono externo ou cone Morse. O objetivo deste estudo foi avaliar comparativamente as características mecânicas de dois tipos de conexões protéticas (hexágono externo e grand Morse) na reabilitação da maxila posterior, empregando o método de elementos finitos: hexágono externo e Morse grande (4,0 mm x 8,0 mm - Hélix Neodent), ambos com coroas retidas por parafuso. Modelos virtuais foram gerados com o software CAD Rhinoceros 7® com base na forma óssea de um segmento da maxila posterior do protocolo BioCAD. Uma malha tetraédrica 3D de ordem primária foi criada para a análise, simulando cargas de 100N a um ângulo de 30º em relação ao eixo do implante. Os resultados mostraram um deslocamento maior no modelo de hexágono externo (0,1399 mm) em comparação ao modelo Morse grande (0,0208 mm). A análise de estresse revelou padrões semelhantes próximos à plataforma do implante, mas o modelo de hexágono externo exibiu maior estresse von Mises (148,4 MPa) em comparação ao modelo Morse grande (99,03 MPa), que apresentou uma melhor dispersão de estresse. O Estresse Principal Máximo foi mais elevado no modelo Morse grande. Assim, o design da conexão da plataforma do implante afeta a distribuição e a intensidade do estresse.
Palavras-chave: Análise por elementos finitos; Prótese dentária fixada por implante; Implante dentário; Maxila.

Resumen
El creciente uso de implantes dentales ha impulsado la demanda de soluciones protésicas innovadoras. Estas nuevas formas y enfoques pretenden mejorar tanto el manejo clínico como la calidad del tratamiento, ofreciendo mejores resultados y una mayor eficiencia en la atención al paciente. Entre los desarrollos recientes, se ha popularizado la conexión grand Morse, pero ningún estudio científico ha comparado su rendimiento mecánico con los implantes de hexágono externo o los implantes de cono Morse. El objetivo de este estudio fue evaluar comparativamente las características mecánicas de dos tipos de conexiones protésicas (hexágono externo y grand Morse) en la rehabilitación del maxilar posterior, utilizando el método de elementos finitos: hexágono externo y grand Morse (4,0 mm x 8,0 mm - Hélix Neodent), ambos con coronas atornilladas. Los modelos virtuales se generaron con el software CAD Rhinoceros 7® basándose en la forma ósea de un segmento maxilar posterior a partir del protocolo BioCAD. Se creó una malla 3D tetraédrica de orden primario para el análisis, simulando cargas de 100 N en un ángulo de 30º respecto al eje del implante. Los resultados mostraron un mayor desplazamiento en el modelo de hexágono externo (0,1399 mm) en comparación con el modelo de gran Morse (0,0208 mm). El análisis de tensiones reveló patrones similares cerca de la plataforma del implante, pero el modelo de hexágono externo mostró una mayor tensión de von Mises (148,4 MPa) en comparación con el modelo grand Morse (99,03 MPa), que presentaba una mejor dispersión de tensiones. La tensión principal máxima fue mayor en el modelo grand Morse. Así pues, el diseño de la conexión de la plataforma del implante afecta a la distribución e intensidad de las tensiones.
Palabras clave: Análisis de elementos finitos; Prótesis dental de soporte implantado; Implante dental; Maxilar superior.

1. Introduction

There has been a significant increase in implant rehabilitations for patients who have experienced the loss of one or more teeth in the past few decades (Alemayehu & Jeng, 2021; Brune et al., 2019; Jafari et al., 2022; Shamami et al., 2014; Yang et al., 2020). Due to the biocompatibility of titanium with adjacent bone tissue, dental implants have achieved high long-term success rates, promoting the process known as osseointegration (Gupta et al., 2021; Jafari et al., 2022; T. Wu et al., 2017). After osseointegration is established, the implant-bone system relies on the biomechanics and chemical stability between the bone tissue and titanium, which must withstand the torque generated by the friction between surfaces during masticatory forces (Jafari et al., 2022).

There is ample evidence in the literature indicating various factors that can contribute to implant failure, including bone quality, surface treatment, geometry, and early bone loss in the peri-implant region (Macedo et al., 2017; Silva et al., 2021). Factors related to the biomechanics of implants are crucial for the success of this type of treatment, as there must be proper dissipation of masticatory forces to the supporting bone (Altıparmak et al., 2023). Therefore, the distribution of stresses, displacements, and deformations in the peri-implant regions has received special attention (Khorshidparast et al., 2023; Nesbitt et al., 2023; Zhang et al., 2022).

One key aspect to consider is the bone stability and preservation around the implant platform (Nesbitt et al., 2023). The type of prosthetic connection present on the platform directly influences the mechanical behavior of the implants and the dissipation of loads to the adjacent bone (Bittencourt et al., 2021; Valera-Jiménez et al., 2020; Vinhas et al., 2020; Zhang et al., 2022). Even with full osseointegration of implants, there remains a risk of bone resorption in the peri-implant regions, often stemming from bacterial infection or excessive loading due to masticatory forces. (Odo et al., 2015; Valera-Jiménez et al., 2020; Vinhas et al., 2020). Therefore, one of the most important variables determining the ability of implants and prosthetic components to tolerate loads during or after osseointegration is the design of the platform (Almeida & Pellizzer, 2008).

Over the last few years, a variety of implant prosthetic connections geometries have emerged, each exhibiting distinct mechanical, biological, and aesthetic characteristics (Almeida & Pellizzer, 2008). These connections primarily fall into two categories: internal and external geometries (Almeida & Pellizzer, 2008). External connections typically feature an external hexagon on the implant platform, whereas internal connections encompass internal hexagons, internal octagons, and Morse taper connections. (Almeida & Pellizzer, 2008; Lemos et al., 2018; Liang et al., 2015). Several studies have presented out mechanical complications with external hexagon connections due to their low resistance to oblique forces (Bittencourt et al., 2021; Menacho-Mendoza et al., 2022; Valera-Jiménez et al., 2020; Vinhas et al., 2020).

Among the reported mechanical problems are screw loosening, screw fracture, and abutment rotation (Odo et al., 2015). For this reason, recent studies have focused on new characteristics of dental implants and prosthetic connection systems, for example the platform switching associated with Morse taper connections (Lemos et al., 2018). Compared to other systems, the Morse taper connection provides greater stability in the crestal bone when subjected to axial and lateral forces (CATÁLOGO DE PRODUTOS NEODENT ® 2 0 2 3, n.d.; H. Wu et al., 2021).

In this context, implants with a new type of prosthetic platform called the grand Morse (GM) have recently been introduced to the market. Similar to Morse taper implants, the GM line also applies the concept of platform switching (Moreira et al., 2022). However, there are currently no studies in the literature that have mechanically tested the effectiveness of this new platform design or confirmed its mechanical similarity to Morse taper implants (Sciasci et al., 2018).

Multiple methods can be utilized to assess the mechanical behavior of an implant-supported prosthesis, such as photoelasticity, holographic interferometry, digital image correlation, and three-dimensional finite element analysis (FEA) (Brozović et al., 2014; Guessasma et al., 2022; Liu et al., 2022; H. Wu et al., 2021). Among these techniques, FEA has demonstrated to be a dependable and precise instrument for examining the mechanical characteristics of implant-supported prostheses and is widely used worldwide in the fields of science and engineering to simulate the behavior of complex systems subjected to loads and deformations (Cervino et al., 2020; Gil-Marques et al., 2022; Roy et al., 2018; Tenenbaum et al., 2003).

FEA has been utilized in the realm of dental implants to forecast stress distribution patterns at the implant-bone interface, encompassing not only shape comparisons. (cylindrical or tapered) (Bordin et al., 2018), diameters (Altıparmak et al., 2023), and lengths (Almeida & Pellizzer, 2008) but also to simulate various oral conditions (Liang et al., 2015) and types of prostheses (Brozović et al., 2014; Buser et al., 2017; Menacho-Mendoza et al., 2022). FEA is a numerical technique enabling stress calculations, displacements, and deformations through the assessment of the evaluation of the equation governing the mechanical properties of materials (Y.-L. Wu et al., 2022). The method offers the advantage of resolving complex structural issues by subdividing intricate geometries into significantly smaller domains (elements) and computing the response to applied forces. (Y.-L. Wu et al., 2022).

The objective of this study was to comparatively assess the mechanical characteristics of two types of prosthetic connections (external hexagon and grand Morse) in posterior maxilla rehabilitation, employing the finite element method.

2. Material and Methods

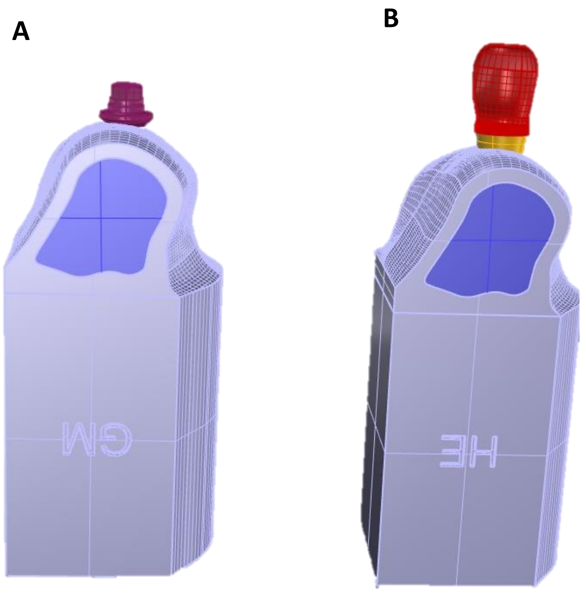
The methodology is important for an experiment to be reproducible. The present study is experimental, laboratory research, of a qualitative and quantitative nature using finite element analysis (Pereira et al., 2018).

Geometry of the models

The methodology is important for an experiment to be reproducible. The present study is experimental, laboratory research, of a qualitative and quantitative nature using finite element analysis (Pereira et al., 2018).

The first step was the generation of CAD drawings of the implants, components, and adjacent bone using Rhinoceros 7® software from McNell. The bone architecture was developed based on the BioCAD protocol. Implants with HE and GM prosthetic connections measuring 4.0 mm x 8.0 mm - Hélix-Neodent were used for the finite element analysis. The prosthetic components used for both analyses were the SF Conical Abutment for HE implants and the Exact GM Abutment for GM implants, both from Neodent®, rehabilitated with a screw-retained device simulating a dental crown. The CAD models of the implants, components, crown, and test bodies are represented in Figure 1A, B.

Figure 1 - CAD of GM and HE implants installed in the bone block. A, lateral view of the GM model CAD, composed of the bone structure and GM abutment (purple). B, lateral view of the HE model CAD, composed of the bone structure, SF conical abutment (yellow), and screw-retained prosthetic crown (red).



Source: Authors' archive

For the finite element analysis, it is necessary to generate a mesh to represent and simplify the model. After editing the CAD files, HyperMesh 2021 software from Altair® was used to define the boundary conditions of the virtual models and create a first-order tetrahedral 3D mesh.

The virtual models represent a posterior segment of the maxilla, specifically in the region of the upper first molar, and simulate trabecular bone covered by a thin layer of 1.5 mm of cortical bone. To build both models, it was necessary to provide the mechanical characteristics of the materials involved in order to calculate the displacement responses and stresses resulting from the applied forces in the analyses. Therefore, it was essential to investigate the literature for values of two mechanical characteristics: the elastic modulus, also known as Young's modulus, and the Poisson's ratio (Table 1).

Table 1 - Values of the mechanical characteristics of the materials used.

Materials	Young's modulus (MPa)	Possion coefficient
Cortical bone (Bittencourt et al., 2021; Patil et al., 2019; Roy et al., 2018; H. Wu et al., 2021; T. Wu et al., 2017)	13.700	0.3
Trabecular bone (Bittencourt et al., 2021; Patil et al., 2019; Roy et al., 2018; H. Wu et al., 2021; T. Wu et al., 2017)	1.370	0.3
Titanium (abutments, screws, implants) (Bittencourt et al., 2021; Patil et al., 2019; Roy et al., 2018; H. Wu et al., 2021; T. Wu et al., 2017)	110.000	0.35
Chorme-cobalt alloy (Bicudo et al., 2016)	218.000	0.33

Source: Authors' archive.

Afterwards the mechanical properties of the materials were determined and inputted into the software, the support and fixation regions were selected, along with the type of contact they exert fixed or sliding. Additionally, the regions for applying the load were defined, including the numerical value and direction. A 100N oblique load was applied to a specific area of the experimental crown structure at a 30° angle relative to the implant's longitudinal axis. Considering the mechanical properties of the materials of the materials and the amount of force applied in the tests, it is assumed that all stresses and displacements were linear, and all materials utilized in the subsequent analyses were considered isotropic.

3. Results

Displacements

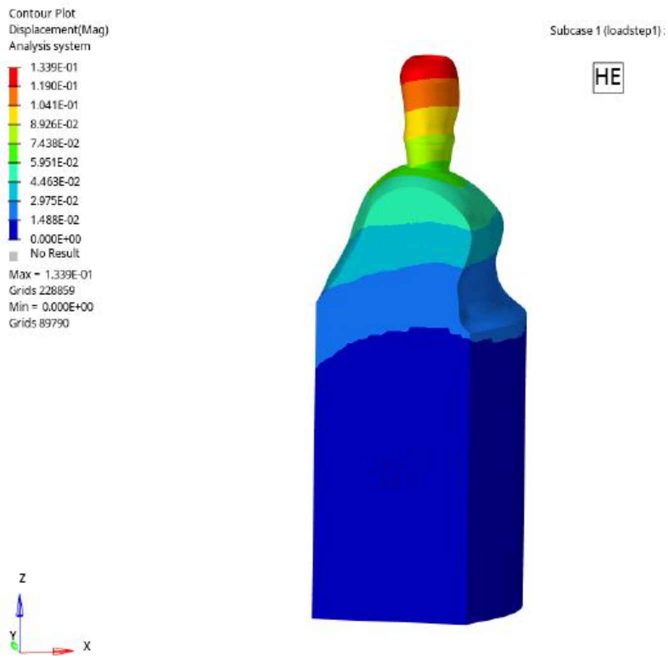
In the displacement analysis, understanding the Cartesian coordinate system being used is essential. In the figures of this section, the XYZ system is presented, where the Y-axis is represented by the green color, the X-axis by the red color, and the Z-axis, finally, by the blue color. They assist in understanding the direction in which the displacement is occurring.

In addition to the coordinate system, understanding the colors on the map is crucial for a good simulation analysis. In the upper-left corner of the figures presented in this section, there is a color scale, ranging from blue (lower) to red (higher). Regions in blue indicate lower displacement, while those in red indicate higher displacement.

Comparatively, a greater displacement was obtained in the EH model, the displacement was recorded as 1.339E-1 (0.1399 mm), whereas in the GM model, under identical conditions, the displacement measured 2.080E-2 (0.0208 mm). The trend of the movement is described in Figures 2 and 3.

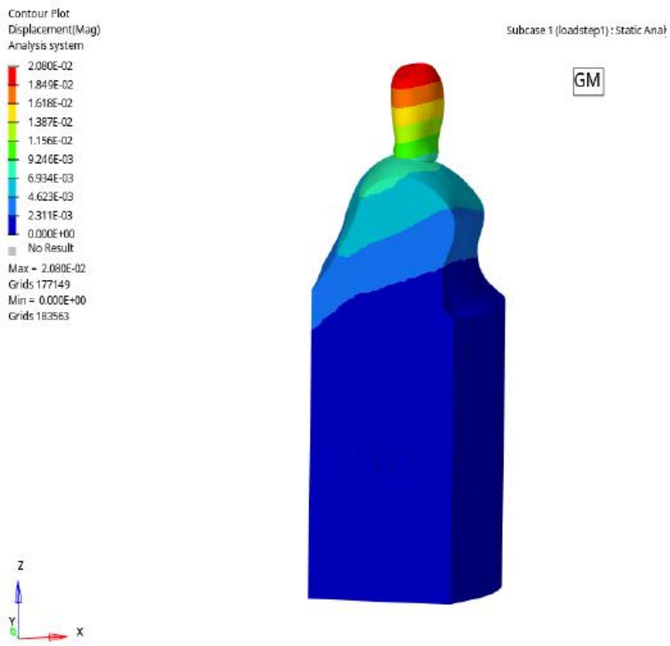
Research, Society and Development, v. 13, n. 10, e53131047055, 2024
(CC BY 4.0) | ISSN 2525-3409 | DOI: http://dx.doi.org/10.33448/rsd-v13i10.47055

Figure 2 - The image displays the total displacements that occurred in the model with the HE implants installed. The red region indicates the area with the highest displacement, predominantly concentrated around the dental crown region.



Source: Authors' archive.

Figure 3 - The image depicts the total displacements that occurred in the model with the GM implant installed. The red region represents the area of the highest displacement and is concentrated in the region of the dental crown.



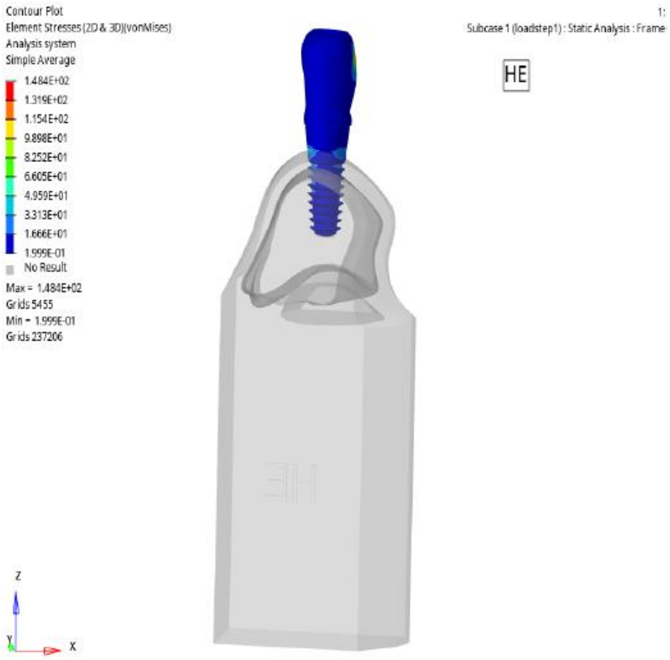
Source: Authors' archive.

Von Mises Stress

When evaluating the model based on the Von Mises stress criterion, which defines stress concentration along the geometry, once again, in the upper-left corner of the figures presented in this section, there is a color scale, ranging from blue (lower) to red (higher). Regions in blue indicate lower stress, while those in red indicate higher stress.

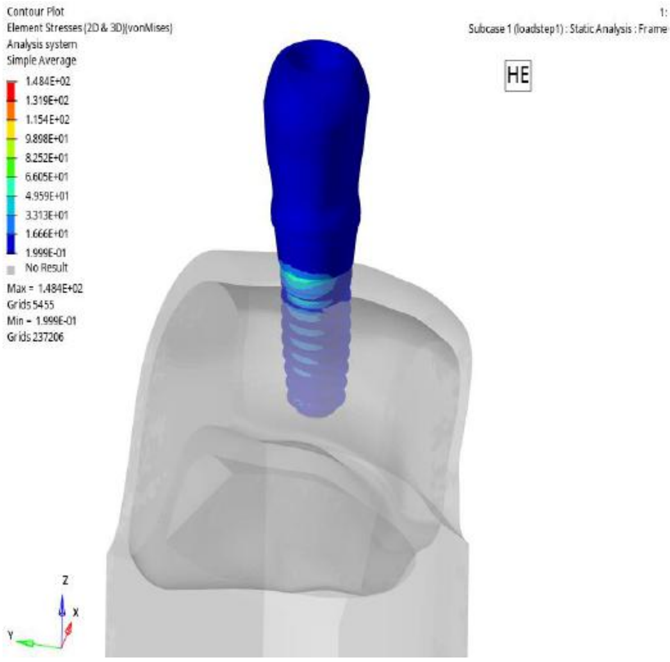
For both models, we observe similar behaviors, with stress accumulation near the implant-bone fixation region, in the direction opposite to the applied force. The EH model exhibits less distributed stress along the implant's long axis and higher intensity, with 148.4 MPa concentrated in the platform region. In contrast, the GM model shows a lower stress of 99.03 MPa in the same region, concentrated on the platform but dissipated into the implant body. These conditions can be observed in Figures 4, 5, 6, and 7.

Figure 4 - Von Mises Stress in the HE model indicates that the highest concentration of stresses occurs in the region of the implant platform.



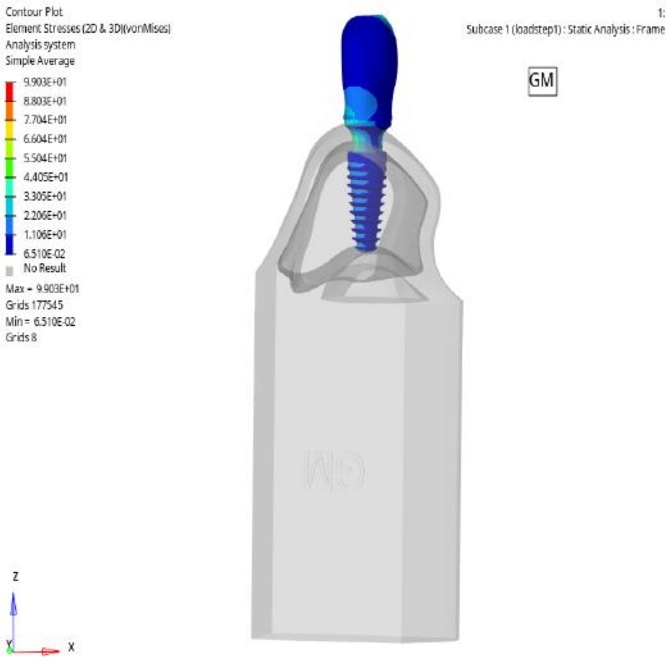
Source: Authors' archive.

Figure 5 - Von Mises Stress in the HE model, as depicted in the detailed view, highlights the highest concentration of stresses in the region of the implant platform.



Source: Authors' archive.

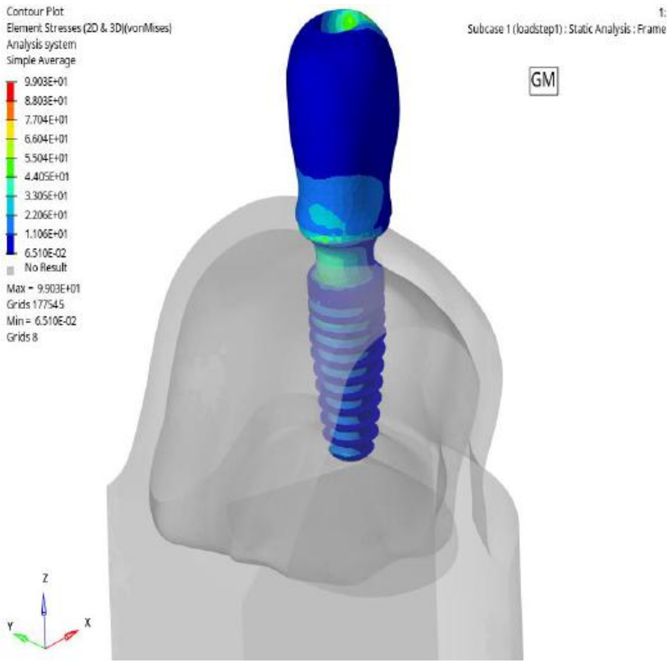
Figure 6 - Von Mises Stress in the GM model reveals that the highest concentration of stresses is present in the region of the implant platform and in the crown of the implant.



Source: Authors' archive.

(CC BY 4.0) | ISSN 2525-3409 | DOI: http://dx.doi.org/10.33448/rsd-v13i10.47055

Figure 7 - Von Mises Stress in the GM model, as observed in the detailed view, demonstrates the highest concentration of stresses in the region of the implant platform and in the crown of the implant.

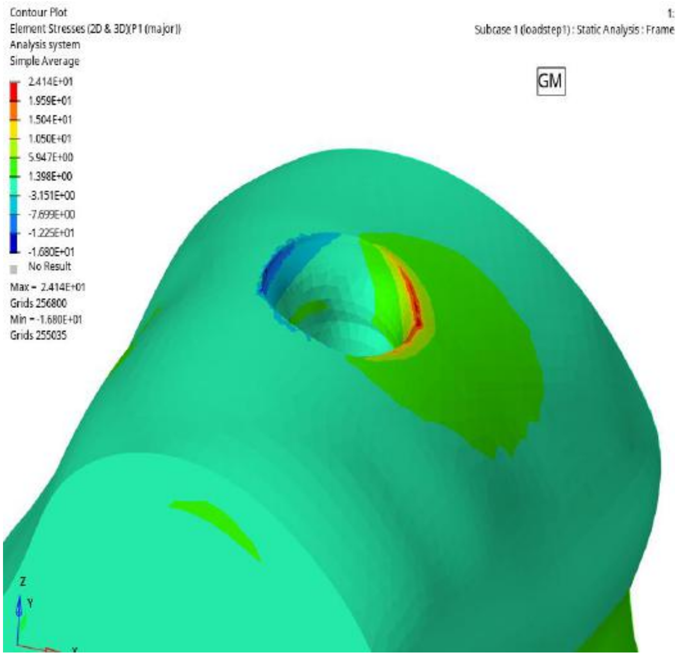


Source: Authors' archive.

Maximum Principal Stress

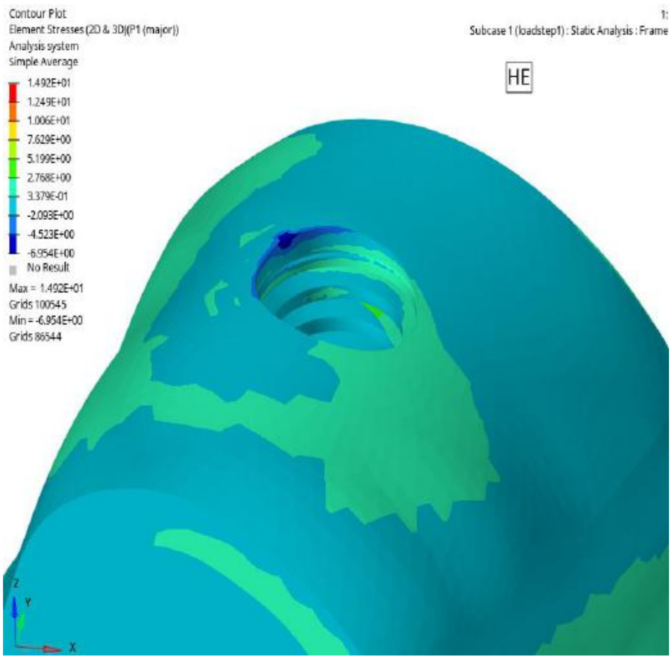
When analyzing stresses within the bone region, the Maximum Principal Stress criterion was used, delineating positive stresses (tension-red) and negative stresses (compression-blue). Using this criterion, A greater concentration of stress in the platform region of the GM model was observed, with two clearly evident regions of tension and compression, with tension values of 24.14 MPa and compression values of -16.80 MPa. In the HE model, the tension region is much less evident, with tension values of 2.76 MPa and compression values of -6.95 MPa, as shown in Figures 8 and 9.

Figure 8 - Maximum Principal Stress in the GM model at the bone-implant interface, illustrating the region where stresses are transferred to the adjacent bone. Regions in red indicate tension zones, and regions in blue indicate compression zones.



Source: Authors' archive.

Figure 9 - Maximum Principal Stress in the HE model at the bone-implant interface, showing the region where stresses are being dissipated to the adjacent bone. Regions in red indicate tension zones, and regions in blue indicate compression zones.



Source: Authors' archive.

(CC BY 4.0) | ISSN 2525-3409 | DOI: http://dx.doi.org/10.33448/rsd-v13i10.47055

4. Discussion

The objective of this study was to evaluate, through the finite element method (FEM), the mechanical behavior of peri-implant bone tissue, implants, and prosthetic components of two different prosthetic connection systems in the posterior maxilla, using short implants with a diameter of 4.0 mm and a length of 8 mm.

Numerical simulation analyses are widely used to understand the stress distribution in implant biomechanics. Three-dimensional finite element analysis allows the observation of stresses caused by occlusal forces in different dental implant treatment simulations. In this study, a three-dimensional finite element analysis was conducted with the application of an oblique load of 100N at a 30º angle to the implant's long axis. Additionally, all materials were considered isotropic and linear, which is a limitation of the analysis.

In this study, the HE implant model exhibited greater displacement in the platform region compared to the GM implant model. This behavior was expected due to the greater lever arm generated by the way HE implants are inserted into the bone. In contrast, the GM model, being more deeply inserted into the bone, showed better fixation and stability, resulting in lower displacements.

Bicudo et al. (2016) compared HE and CM platforms through fatigue cycles and finite element analysis, obtaining lower displacements for the CM system. They found that the CM system was less sensitive to load variations, attributing its performance to the locking mechanism between the implant and the component, reducing vibrations and micromovements.

The von Mises stress criterion measures stresses concentrated in the metal structures of the model and their distribution among the components. In the HE model, von Mises stresses were higher than in the GM model. The HE model also exhibited stress concentration on the platform and less stress dissipation along the implant's long axis, while the GM model showed lower stress in the platform region, less concentrated and dissipated along the implant's long axis. This difference may be attributed to the intraosseous position of the GM implant, providing greater stability and anchorage to the supporting bone and preventing micromovements.

Macedo et al. (2017) also compared CM and HE connection implants using finite element methods under oblique loads. Similar to the current study, they obtained higher von Mises stresses for HE implants compared to CM implants.

The Maximum Principal Stress criterion measures stresses generated in the adjacent bone tissue. Unlike the other analyses, the GM model showed higher stress values in the adjacent bone than the HE model. This can be explained by the greater stability and anchorage of the GM model to the supporting bone, leading to a higher transfer of stresses to the bone. The GM model exhibited two clearly evident regions of tension and compression, different from the HE model. It is important to note that, despite transferring more stress, GM model values do not exceed physiological limits of bone tissue, and they can even be beneficial for the osseointegration process and bone remodeling (Lemos et al., 2021). On the other hand, the HE model concentrates all stresses between the implant and the prosthetic component, facilitating screw fractures and pillar displacements, which can be detrimental in clinical practice (Sciasci et al., 2018).

Finite element analysis enables the prediction of stress distribution around implants in both the cortical bone region and the more apical area involving trabecular bone. This technique can anticipate the mechanical performance of various implants concerning their design, load, bone quality, and materials. In this study, a detailed evaluation of stress magnitude around implants with two distinct platform designs, GM and HE, was conducted.

As evidenced, the use of CM connection implants represents a successful approach to reduce bone loss around implants. The deeper the prosthetic connection within the implant body, the more evenly occlusal loads are dissipated, as stresses are distributed along the implant's wall and, consequently, throughout the implant, rather than being concentrated in the platform region (Lemos et al., 2018).

Following the same characteristics and objectives as CM, the GM connection was developed. The GM connection has an internal angle of 16°, and its thicker internal walls contribute to greater mechanical resistance and better results. However, there are currently no studies comparing the effectiveness of this new connection type (Moreira et al., 2022). Within the limitations of this study, the results are consistent with those reported in the literature regarding stress and displacement criteria in CM and HE implant systems, suggesting that GM implants exhibited the same mechanical behavior as CM implants. It is important to note that there are no mechanical studies proving that GM modifications are superior or equal to CM, as they are different implants with different components.

5. Conclusion

Within the constraints of this study, a comprehensive assessment of stress and displacement magnitude around peri-implant bone was achieved for two distinct implant systems: external hexagon (HE) and grand Morse (GM). It can be inferred that the prosthetic connection design of the implant significantly influences the distribution and intensity of stress and displacements in the prosthesis-implant system. GM implants demonstrated a favorable mechanical behavior, mirroring the pattern observed in CM implants. Future studies could evaluate different types of masticatory load and variations in bone density in order to better investigate the distribution of stresses and displacements in the peri-implant bone. Furthermore, we recommend investigating other prosthetic connections to broaden the applicability of the results. More detailed simulations, considering specific clinical conditions and long-term analyses, as well as experimental studies that complement the numerical analyses, could strengthen the basis for clinical practice and validate the findings of this study.

Acknowledgments

The authors would like to express their gratitude to Neodent for providing all the implants and prosthetic components used in the study, to the Technology Information Center - Renato Archer, for developing the finite element analyses, and to CAPES for the financial support that enabled the completion of this work. Financial support provided by CAPES-88887.570438/2020-00.

Conflict of Interest

The authors declare no conflict of interest.

References

Alemayehu, D.-B., & Jeng, Y.-R. (2021). Three-Dimensional Finite Element Investigation into Effects of Implant Thread Design and Loading Rate on Stress Distribution in Dental Implants and Anisotropic Bone. *Materials (Basel, Switzerland)*, *14*(22). https://doi.org/10.3390/ma14226974.

Almeida, E. O. De, & Pellizzer, E. P. (2008). Biomecânica em prótese sobre implante relacionada às inclinações das cúspides e às angulações dos implantes osseointegrados-revisão de literatura. *Revista de Odontologia Da UNESP*, *37*(4), 321–327.

Altıparmak, N., Polat, S., & Onat, S. (2023). Finite element analysis of the biomechanical effects of titanium and Cfr-peek additively manufactured subperiosteal jaw implant (AMSJI) on maxilla. *Journal of Stomatology, Oral and Maxillofacial Surgery, 124*(1S), 101290. https://doi.org/10.1016/j.jormas.2022.09.011.

Bicudo, P., Reis, J., Deus, A. M., Reis, L., & Vaz, M. F. (2016). Performance evaluation of dental implants: An experimental and numerical simulation study. *Theoretical and Applied Fracture Mechanics*, *85*, 74–83. https://doi.org/10.1016/j.tafmec.2016.08.014.

Bittencourt, A. B. B. C., de Moraes Melo Neto, C. L., Penitente, P. A., Pellizzer, E. P., Santos, D. M. Dos, & Goiato, M. C. (2021). Comparison of the Morse Cone Connection with the Internal Hexagon and External Hexagon Connections Based on Microleakage - Review. *Prague Medical Report*, *122*(3), 181–190. https://doi.org/10.14712/23362936.2021.15.

Bordin, D., Witek, L., Fardin, V. P., Bonfante, E. A., & Coelho, P. G. (2018). Fatigue Failure of Narrow Implants with Different Implant-Abutment Connection Designs. *Journal of Prosthodontics*, *27*(7), 659–664. https://doi.org/10.1111/jopr.12540.

Brozović, J., Demoli, N., Farkaš, N., Sušić, M., Alar, Z., & Pandurić, D. G. (2014). Properties of axially loaded implant-abutment assemblies using digital holographic interferometry analysis. *Dental Materials : Official Publication of the Academy of Dental Materials*, *30*(3), e17-27. https://doi.org/10.1016/j.dental.2013.12.005.

Brune, A., Stiesch, M., Eisenburger, M., & Greuling, A. (2019). The effect of different occlusal contact situations on peri-implant bone stress - A contact finite element analysis of indirect axial loading. *Materials Science & Engineering. C, Materials for Biological Applications*, *99*, 367–373. https://doi.org/10.1016/j.msec.2019.01.104.

Buser, D., Sennerby, L., & Bruyn, H. De. (2017). Modern implant dentistry based on osseointegration: 50 years of progress, current trends and open questions. *Periodontology 2000*, *73*(1), 7–21. https://doi.org/10.1111/prd.12185.

CATÁLOGO DE PRODUTOS NEODENT ® 2 0 2 3. (n.d.).

Cervino, G., Fiorillo, L., Arzukanyan, A. V, Spagnuolo, G., Campagna, P., & Cicciù, M. (2020). Application of bioengineering devices for stress evaluation in dentistry: the last 10 years FEM parametric analysis of outcomes and current trends. *Minerva Stomatologica*, *69*(1). https://doi.org/10.23736/S0026-4970.19.04263-8.

Gil-Marques, B., Pallarés-Sabater, A., Brizuela-Velasco, A., Lasheras, F. S., Lázaro-Calvo, P., Gómez-Adrián, M. D., & Larrazábal-Morón, C. (2022). A Biomechanical Analysis of the Influence of the Morfology of the Bone Blocks Grafts on the Transfer of Tension or Load to the Soft Tissue by Means of the Finite Elements Method. *Materials*, *15*(24), 9039. https://doi.org/10.3390/ma15249039.

Guessasma, S., Nouri, H., & Belhabib, S. (2022). Digital Image Correlation and Finite Element Computation to Reveal Mechanical Anisotropy in 3D Printing of Polymers. *Materials (Basel, Switzerland)*, *15*(23). https://doi.org/10.3390/ma15238382.

Gupta, Y., Iyer, R., Dommeti, V. K., Nutu, E., Rana, M., Merdji, A., Biswas, J. K., & Roy, S. (2021). Design of dental implant using design of experiment and topology optimization: A finite element analysis study. *Proceedings of the Institution of Mechanical Engineers. Part H, Journal of Engineering in Medicine*, *235*(2), 157–166. https://doi.org/10.1177/0954411920967146.

Jafari, B., Katoozian, H. R., Tahani, M., & Ashjaee, N. (2022). A comparative study of bone remodeling around hydroxyapatite-coated and novel radial functionally graded dental implants using finite element simulation. *Medical Engineering & Physics*, *102*, 103775. https://doi.org/10.1016/j.medengphy.2022.103775.

Khorshidparast, S., Akhlaghi, P., Rouhi, G., & Barikani, H. (2023). Measurement of bone damage caused by quasi-static compressive loading-unloading to explore dental implants stability: Simultaneous use of in-vitro tests, μ-CT images, and digital volume correlation. *Journal of the Mechanical Behavior of Biomedical Materials*, *138*, 105566. https://doi.org/10.1016/j.jmbbm.2022.105566.

Lemos, C. A. A., Verri, F. R., Noritomi, P. Y., Kemmoku, D. T., de Souza Batista, V. E., Cruz, R. S., de Luna Gomes, J. M., & Pellizzer, E. P. (2021). Effect of bone quality and bone loss level around internal and external connection implants: A finite element analysis study. *The Journal of Prosthetic Dentistry*, *125*(1), 137.e1-137.e10. https://doi.org/10.1016/j.prosdent.2020.06.029.

Lemos, C. A. A., Verri, F. R., Santiago, J. F., de Faria Almeida, D. A., de Souza Batista, V. E., Noritomi, P. Y., & Pellizzer, D. P. (2018). Retention System and Splinting on Morse Taper Implants in the Posterior Maxilla by 3D Finite Element Analysis. *Brazilian Dental Journal*, *29*(1), 30–35. https://doi.org/10.1590/0103-6440201801492.

Liang, R., Guo, W., Qiao, X., Wen, H., Yu, M., Tang, W., Liu, L., Wei, Y., & Tian, W. (2015). Biomechanical analysis and comparison of 12 dental implant systems using 3D finite element study. *Computer Methods in Biomechanics and Biomedical Engineering*, *18*(12), 1340–1348. https://doi.org/10.1080/10255842.2014.903930.

Liu, B., Xu, W., Chen, M., Chen, D., Sun, G., Zhang, C., Pan, Y., Lu, J., Guo, E., & Lu, X. (2022). Structural Design and Finite Element Simulation Analysis of Grade 3 Graded Porous Titanium Implant. *International Journal of Molecular Sciences*, *23*(17). https://doi.org/10.3390/ijms231710090.

Macedo, J. P., Pereira, J., Faria, J., Pereira, C. A., Alves, J. L., Henriques, B., Souza, J. C. M., & López-López, J. (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.

Menacho-Mendoza, E., Cedamanos-Cuenca, R., & Díaz-Suyo, A. (2022). Stress analysis and factor of safety in three dental implant systems by finite element analysis. *The Saudi Dental Journal*, *34*(7), 579–584. https://doi.org/10.1016/j.sdentj.2022.08.006.

Moreira, G. M., de Mattos Peres, G., & dos Reis, T. A. (2022). Diferentes sistemas de implantes dentários: uma revisão descritiva da literatura. *Research, Society and Development*, *11*(8), e16311830603. https://doi.org/10.33448/rsd-v11i8.30603.

Nesbitt, D. Q., Burruel, D. E., Henderson, B. S., & Lujan, T. J. (2023). Finite element modeling of meniscal tears using continuum damage mechanics and digital image correlation. *Scientific Reports*, *13*(1), 4039. https://doi.org/10.1038/s41598-023-29111-z.

Odo, C. H., Pimentel, M. J., Consani, R. L. X., Mesquita, M. F., & Nóbilo, M. A. A. (2015). Stress on external hexagon and Morse taper implants submitted to immediate loading. *Journal of Oral Biology and Craniofacial Research*, *5*(3), 173–179. https://doi.org/10.1016/j.jobcr.2015.07.002.

Patil, S. M., Deshpande, A. S., Bhalerao, R. R., Metkari, S. B., & Patil, P. M. (2019). A three-dimensional finite element analysis of the influence of varying implant crest module designs on the stress distribution to the bone. *Dental Research Journal*, *16*(3), 145–152.

Pereira A. S. et al. (2018). Metodologia da pesquisa científica. [free e-book]. Santa Maria/RS. Ed. UAB/NTE/UFSM.

Roy, S., Dey, S., Khutia, N., Chowdhury, A. R., & Datta, S. (2018). Design of patient specific dental implant using FE analysis and computational intelligence techniques. *Applied Soft Computing*, *65*, 272–279. https://doi.org/10.1016/j.asoc.2018.01.025.

Sciasci, P., Casalle, N., & Vaz, L. G. (2018). Evaluation of primary stability in modified implants: Analysis by resonance frequency and insertion torque. *Clinical Implant Dentistry and Related Research*, *20*(3), 274–279. https://doi.org/10.1111/cid.12574.

Shamami, D. Z., Karimi, A., Beigzadeh, B., Derakhshan, S., & Navidbakhsh, M. (2014). A Three-Dimensional Finite Element Study to Characterize the Influence of Load Direction on Stress Distribution in Bone Around Dental Implant. *Journal of Biomaterials and Tissue Engineering*, *4*(9), 693–699. https://doi.org/10.1166/jbt.2014.1230.

Silva, G. A. F., Faot, F., da Rosa Possebon, A. P., da Silva, W. J., & Cury, A. A. D. B. (2021). Effect of macrogeometry and bone type on insertion torque, primary stability, surface topography damage and titanium release of dental implants during surgical insertion into artificial bone. *Journal of the Mechanical Behavior of Biomedical Materials*, *119*, 104515. https://doi.org/10.1016/j.jmbbm.2021.104515.

Tenenbaum, H., Schaaf, J.-F., & Cuisinier, F. J. G. (2003). Histological Analysis of the Ankylos Peri-implant Soft Tissues in a Dog Model. *Implant Dentistry*, *12*(3), 259–265. https://doi.org/10.1097/01.ID.0000075720.78252.54.

Valera-Jiménez, J. F., Burgueño-Barris, G., Gómez-González, S., López-López, J., Valmaseda-Castellón, E., & Fernández-Aguado, E. (2020). Finite element analysis of narrow dental implants. *Dental Materials : Official Publication of the Academy of Dental Materials*, *36*(7), 927–935. https://doi.org/10.1016/j.dental.2020.04.013.

Vinhas, A. S., Aroso, C., Salazar, F., López-Jarana, P., Ríos-Santos, J. V., & Herrero-Climent, M. (2020). Review of the Mechanical Behavior of Different Implant-Abutment Connections. *International Journal of Environmental Research and Public Health*, *17*(22). https://doi.org/10.3390/ijerph17228685.

Wu, H., Shi, Q., Huang, Y., Chang, P., Huo, N., Jiang, Y., & Wang, J. (2021). Failure Risk of Short Dental Implants Under Immediate Loading: A Meta-Analysis. *Journal of Prosthodontics : Official Journal of the American College of Prosthodontists*, *30*(7), 569–580. https://doi.org/10.1111/jopr.13376.

Wu, T., Fan, H., Ma, R., Chen, H., Li, Z., & 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 & Engineering. C, Materials for Biological Applications*, *75*, 297–304. https://doi.org/10.1016/j.msec.2016.11.041.

Wu, Y.-L., Tsai, M.-H., Chen, H.-S., Lin, C.-P., & Wu, A. Y.-J. (2022). Effect of Marginal Bone Integrity and Aftermarket Abutment Screws on Dental Implant Systems-A Preliminary Study with Finite Element Method. *Materials (Basel, Switzerland)*, *15*(17). https://doi.org/10.3390/ma15175952.

Yang, B., Irastorza-Landa, A., Heuberger, P., & Ploeg, H.-L. (2020). Effect of insertion factors on dental implant insertion torque/energy-experimental results. *Journal of the Mechanical Behavior of Biomedical Materials*, *112*, 103995. https://doi.org/10.1016/j.jmbbm.2020.103995.

Zhang, W.-T., Cheng, K.-J., Liu, Y.-F., Wang, R., Chen, Y.-F., Ding, Y., Yang, F., & Wang, L.-H. (2022). Effect of the prosthetic index on stress distribution in Morse taper connection implant system and peri-implant bone: a 3D finite element analysis. *BMC Oral Health*, *22*(1), 431. https://doi.org/10.1186/s12903-022-02465-y.