A new approach to evaluate the biomechanical characteristics of osseointegrated dental implants

V. Ricotta*, T. Ingrassia, V. Nigrelli, M. Zicari

Università degli Studi di Palermo, Dipartimento di Ingegneria, Viale delle Scienze – 90128 Palermo, Italy,

* Corresponding author. Tel.:+39 091 23897263; E-mail address: vito.ricotta@unipa.it

Abstract. Tooth loss is a common pathology that affects many people. Dental osseointegrated implants are the ideal solution to restore normal functionality in partially or completely edentulous patients. The not perfect osseointegration and the fixture fracture are the main causes of failure for these kinds of implant. To avoid these drawbacks, several studies have been conducted to analyse the behaviour of dental implants.

Aim of this work is to analyse the biomechanical behaviour of three different endosseous dental implants. For this purpose, a new numerical model has been developed to simulate different levels of osseointegration and to evaluate the stress values on the bone at different times. In this way, it can be investigated the possibility of anticipating the use of dental implants that usually is delayed three months after surgery. Obtained results confirm the validity of the proposed approach and can provide useful guidelines for dentists.

Keywords: Dental implant, Osseointegration, FEM, Virtual simulation, CAD

1 Introduction

Periodontitis, caries, trauma, developmental defects and genetic disorders represent the most common causes of tooth loss [1].

Before dental implants were introduced, dentures or bridges were used to treat these pathologies in edentulous patients. Unfortunately, these solutions did not guarantee high performances, stability and duration.

Nowadays, dental implants are the ideal solution for lack of dentition, being considered the best alternative to natural teeth [2,3].

It is possible to use dental implants to substitute single or multiple teeth. As a single-tooth replacement, an implant is freestanding, not attached to adjacent teeth and, therefore, much easier to maintain. For patients who have lost multiple teeth, dental implants can provide support for a stable, healthy and aesthetic dentition that is not removable. Perhaps the most important aspect of implants is that they can restore a high quality of life and greatly boost a patient's self-esteem.

Based on their structural and functional characteristics, dental implants can be classified as:

- *endosseous implants*: having a pin (fixture) deeply inserted into the bone tissue (also called "root-form");
- *subperiosteal implants*: used when the width and depth of the bone are not such as to allow the insertion of endosseous pins. They are designed to be inserted on the upper part of the bone, but always under the gum;
- *transosseous implants*: with this type of implant, it is possible to treat patients in which bone atrophy is particularly severe without resorting to invasive surgeries (with staples, single pivot, and needle).

Today, the root-form is the most frequently used dental implant. These kinds of implants consist of three main components: fixture, abutment and prosthesis (Fig. 1).

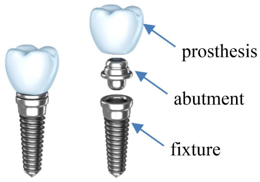

Fig. 1. A root-form implant and its main components.

The fixture is a cylinder (or cone)-shaped metal pin that is surgically embedded into the bone (jaw or mandible). The abutment is placed above the mucosal surface and is attached to the fixture using a screw. The prosthesis is fixed to the abutment either through cement or by means of a screw [1].

A good osseointegration, defined as the direct structural and functional connection between the bone and the implant [4,5], guarantees the implant stability. Two kinds of stability could be considered in dental implants. Primary stability characterizes the mechanical engagement of the implant just after its insertion, while secondary stability is the result of (long-term) bone regeneration and remodelling (biological process) around the inserted implant. Primary and secondary stabilities are closely related [6,7]. Usually, failure of dental implants occurs due to not perfect implant osseointegration or fixture fracture [5,8-10].

To reduce the number of implant failure, several studies have been conducted to analyse the behaviour of dental implants [11-14]. In this field, the numerical methods are widely used [15-16] for the investigation of the biomechanical behaviour of bone-implant components and simulating their mechanical interaction [17-24].

Aim of this work is to study the biomechanical behaviour of three different endosseous dental implants and the development of a new parametric numerical model [17,25] to simulate different levels of osseointegration in order to evaluate the stress distribution on the bone-implant interface at different times, also to evaluate an early use of the dental implants [26,27].

2 CAD modelling

To setup FEM analyses of the dental implants, the 3D models of a mandible and the three types of analysed fixtures have been created by means of reverse engineering tools and 3D modelling parametric software.

2.1 CAD modelling of the fixtures

Three different types of fixture have been studied: a conical-shaped with triangular thread, a cylindrical-shaped with square thread and a cylindrical-shaped with triangular thread (Fig. 2). The main dimensions of the fixtures have been obtained from scientific literature and catalogues. All three types of analysed fixtures have the same maximum diameter and the same length.

Fig. 2. CAD models of three different fixtures: conical with triangular thread (left), cylindrical with square thread (centre) and cylindrical with triangular thread (right).

2.2 3D acquisition and CAD modelling of the mandible

The CAD model of a mandible has been created using DICOM images of computerized tomography (CT) scan data. The 3D model has been reconstructed by semi-automatic segmentation procedures using the commercial medical imaging software Mimics. The final 3D solid model of the mandible is shown in figure 3.

To reduce the computational costs of the FEM analyses, it was decided to study only a limited part of the mandible. For this reason, the 3D model of the bone has been sectioned in the area where the dental implant is installed (Fig. 3).

Fig. 3. CAD model of the mandible.

The mandibular bone has been subdivided in two main regions: a 2 mm thick (external) part of cortical bone and an internal one made of cancellous bone (Fig. 4). The separation surface between cortical and cancellous regions has been obtained by imposing an offset to the external surface.

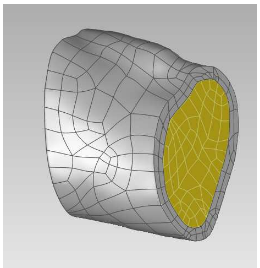

Fig. 4. CAD model of the analysed portion of the mandible: cortical bone (in grey) and cancellous one (in yellow).

Finally, the assemblies of the CAD models of the bone and the fixtures have been made following the surgical guidelines typical of dental surgery (Fig. 5). The interfaces among the different objects have been modelled by boolean operations to make them perfectly complementary.

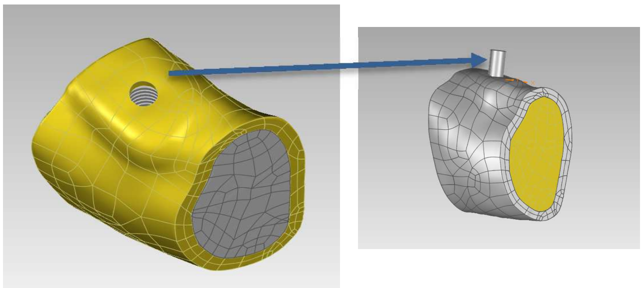

Fig. 5. CAD model of a dental implant-bone assembly.

3 FEM analyses

FEM analyses have been performed to study the biomechanical behaviour of three different endosseous dental implants.

Two different forces [17,26] have been applied to the implant: a compression load of 400 N and a shear load of 150 N distributed on the surface of the upper surface of the fixture (Fig. 6).

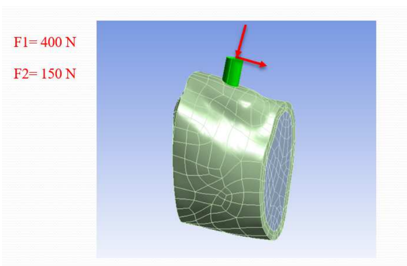

Fig. 6. Applied loads.

After a convergence analysis, it was chosen to use about 1.000.000 tetrahedral elements to mesh the implant-bone assembly (Fig. 7). To improve the quality of the mesh, the crests of the cylindrical fixtures with triangular thread have been truncated.

To obtain very accurate results also in critical areas, the mesh of the assembly has been refined at the screw-bone interfaces and where small curvatures of the geometry occur.

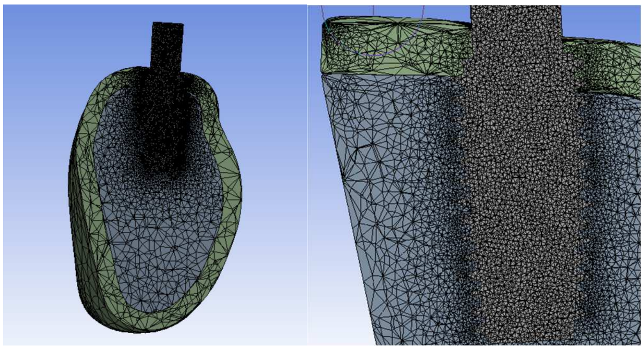

Fig. 7. Mesh of model.

Two fixed constraints [28] have been applied to the extreme surfaces of the analysed portion of the mandible. The contact between the bone and the implant has been considered as bonded.

The material used for the fixture is the titanium alloy Ti6Al4V; the bone structure was considered isotropic (both for cortical and spongy bone) as reported in [29-31]. Table 1 shows the main mechanical properties of the used materials.

Table 1. Mechanical properties of the materials.

Materials	Young modulus [MPa]	Poisson ratio
Ti6Al4V	110000	0.3
Cortical bone	13400	0.3
Cancellous bone	1370	0.31

Results

Figures 8, 9 and 10 show the Von Mises stress maps on the bone respectively for the conical fixture with triangular thread and the cylindrical fixtures with square and triangular thread.

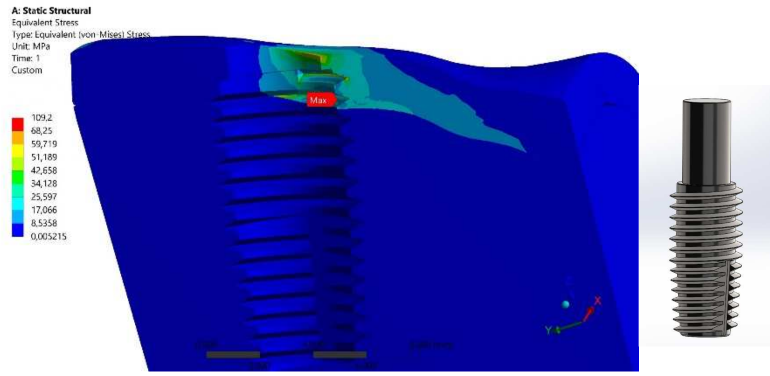

Fig. 8. Conical fixture with triangular thread: Von Mises stress map.

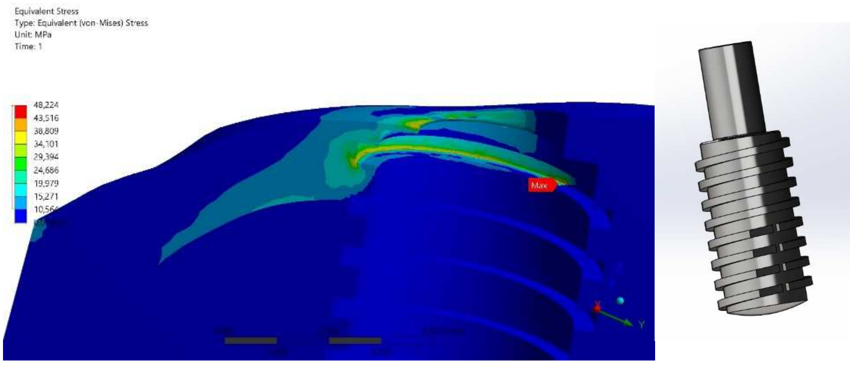

Fig. 9. Cylindrical fixture with square thread: Von Mises stress map.

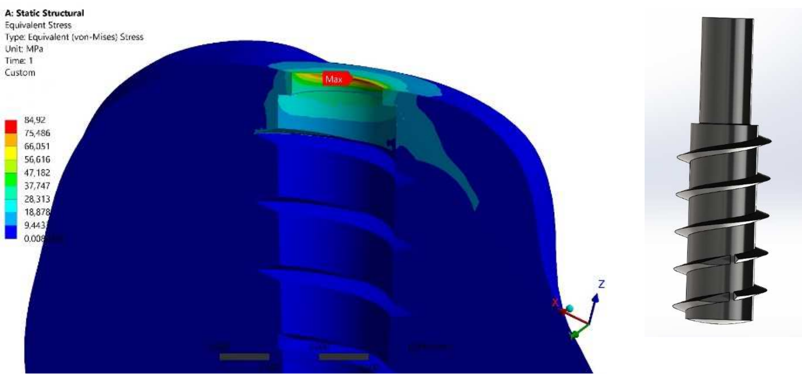

Fig. 10. Cylindrical fixture with triangular thread: Von Mises stress map.

The results of the three FEM simulations show the most stressed parts are the cortical areas around the screw and the cancellous bone-screw interfaces.

The lowest and highest values of von Mises stress have been calculated respectively for the cylindrical fixture with square thread and the conical fixture with triangular thread. In all the cases analysed, the maximum von Mises stresses are lower than the ultimate tensile stress of the cortical bone (150 MPa).

4 New numerical model to simulate different levels of osseointegration

To evaluate the von Mises stress distributions at different times after the surgery, a new parametric numerical model [17,25] has been developed. In particular, the solid model of the bone-implant interface has been suitably subdivided into two different cylindrical regions (“inner” and “outer”) to which different mechanical characteristics can be assigned (Fig. 11).

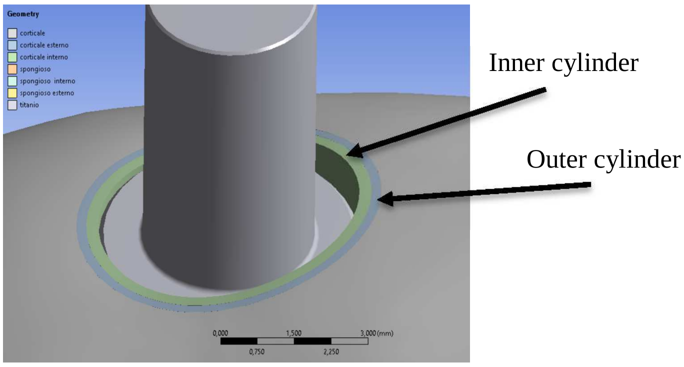

Fig. 11. FEM model with two cylindrical parts having different mechanical characteristics.

By assigning different and increasing mechanical characteristics (in terms of Young modulus) to these cylindrical parts, different levels of osseointegration can be simulated. In particular, lower values of the Young modulus can be used to simulate the initial level of osseointegration, just after the surgery, when the mechanical characteristics at the bone-implant interface are poor. Higher and higher values of the Young modulus can be used to simulate successive levels of osseointegration, until the complete osseointegration is obtained (about 90 days after the surgery), when good mechanical properties occur because the bone is fully reconstructed. To simulate three different phases after the surgery, corresponding to the initial, intermediate and final levels of osseointegration, the mechanical characteristics shown in table 2 have been assigned to the cylindrical parts and three FEM simulations have been performed using these values for cortical and cancellous bones.

Table 2. Mechanical properties of the cylindrical parts at different levels of osseointegation.

Level of osseointegration	Initial (just after surgery)	Intermediate (after 45 days from surgery)	Final (after 90 days from surgery)
Inner cylinder - Cortical bone: value of the Young modulus [MPa]	670 (5% of the final value)	6700 (50% of the final value)	13400
Inner cylinder - Cancellous bone: value of the Young modulus [MPa]	68 (5% of the final value)	685 (50% of the final value)	1370
Outer cylinder - Cortical bone: value of the Young modulus [MPa]	6700 (50% of the final value)	10720 (80% of the final value)	13400
Outer cylinder - Cancellous bone: value of the Young modulus [MPa]	685 (50% of the final value)	1096 (80% of the final value)	1370

The developed numerical model has been tested with the cylindrical fixture with square thread, for which the lowest stress values were calculated during the previous numerical simulations.

Results

Figure 12 shows the Von Mises stress map for the first level of osseointegration. In this case, it has been simulated the effect of the loading just after the surgery. As it can be noticed, the maximum value of the Von Mises stress is 136 MPa and the most stressed area is the cortical area.

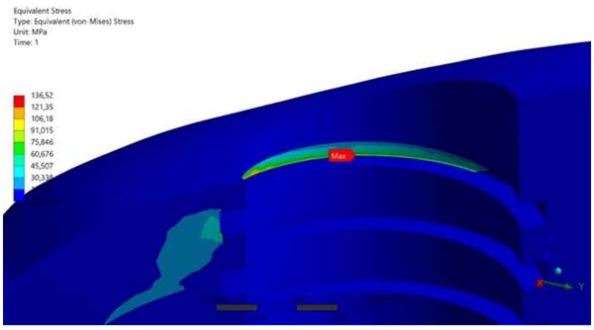

Fig. 12. Cylindrical fixture with square thread: Von Mises stress map just after the surgery.

Figure 13, instead, shows Von Mises stress map at an intermediate level of osseointegration (about 45 days after surgery). The maximum value of the Von Mises stress is 92 MPa.

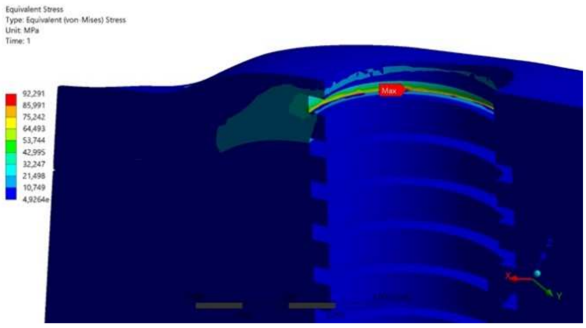

Fig. 13. Cylindrical fixture with square thread: Von Mises stress map 45 days after the surgery.

The obtained results demonstrate that it could be evaluated anticipating the use of the dental implant. Considering the ultimate tensile strength of the cortical bone is equal to 150 MPa [5] and that after 45 days from the surgery the maximum von Mises stress calculated on the cortical bone is about 92 MPa, the dental implant could be used quite safely.

5 Conclusions

In this work the study of the biomechanical behaviour of three different endosseous dental implants and a new numerical model to simulate different levels of osseointegration have been presented.

The first step of the work was related to the modelling of the mandible and the fixtures of the analysed dental implants. After, the CAD models of the mandible and the dental implants have been virtually assembled following, step-by-step, the surgical guidelines.

FEM analyses performed on the three analysed endosseous dental implants have demonstrated that the best results, in terms of lowest von Mises stress calculated on the bone, are obtained for the cylindrical fixture with square thread.

The new numerical model to simulate different levels of osseointegration has been tested on the cylindrical fixture with square thread. The obtained results have demonstrated that it could be possible to evaluate anticipating the use of this kind of dental implant of 45 days.

The developed FEM models could be used also customizing the values of the mechanical characteristics of the bone depending on the specific patients. The imposed mechanical characteristics, in fact, could be defined depending of the porosity of the bone. All that allows giving accurate guidelines for dentists during rehabilitation of patient, allowing to reduce the time between the surgery and the full use of the implants.

References

1. Vahe M.Zohrabian, Michael Sonick, Debby Hwang, James J. Abrahams: Dental implants. Semin Ultrasound CT and MRI 36:415-426 (2015).
2. Michael S. Block: Dental Implants: The Last 100 Years. American Association of Oral and Maxillofacial Surgeons J Oral Maxillofac Surg 76:11-26, (2018).
3. P. Bicudo, J. Reis, A.M. Deus, L. Reis, M. F. Vaz: Mechanical behaviour of dental implants. Procedia Structural Integrity 1 026–033 (2016).
4. Furqan A. Shah, Peter Thomsen, Anders Palmquist: Osseointegration and current interpretations of the bone-implant interface. Acta Biomaterialia 84 1–15 (2019).
5. Mirulla, A.I., Bragonzoni, L., Zaffagnini, S., Bontempi, M., Nigrelli, V., Ingrassia, T., Virtual simulation of an osseointegrated trans-humeral prosthesis: A falling scenario, 2018, Injury, 49 (4), pp. 784-791.
6. A. Dorogoy, D. Rittel, K. Shemtov-Yona, R. Korabi: Modeling dental implant insertion. Journal of the mechanical behavior of biomedical materials 68 42–50 (2017).
7. Gustav Sundell, Christer Dahlin, Martin Andersson, Mattias Thuvander: The bone-implant interface of dental implants in humans on the atomic scale. Acta Biomaterialia 48 445–450 (2017).
8. Sorin Mihai Croitoru, Ion Alexandru Popovici: R&D on dental implants breakage. Applied Surface Science 417 262–268 (2017).
9. Kevin Liaw, Ronald H.Delfini, JamesJ.Abrahams: Dental Implant Complications. Seminars in Ultrasound CT and MRI 36:427-433 (2015).
10. Reza Tabrizi, Hossein Behnia, Shahram Taherian, Nima Hesami: What Are the Incidence and Factors Associated With Implant Fracture? American Association of Oral and Maxillofacial Surgeons J Oral Maxillofac Surg 75:1866-1872, (2017).
11. P. Bicudo, J. Reis, A.M. Deus, L. Reis, M.F. Vaz: Performance evaluation of dental implants: An experimental and numerical simulation study. Theoretical and Applied Fracture Mechanics 85 74–83 (2016).
12. Sandipan Roy, Swati Dey, Niloy Khutia, Amit Roy Chowdhury, Shubhabrata Datta: Design of patient specific dental implant using FE analysis and computational intelligence techniques. Applied Soft Computing 65 272–279 (2018).
13. Kun Wang, Jianping Geng, David Jones, Wei Xu: Comparison of the fracture resistance of dental implants with different abutment taper angles. Materials Science and Engineering C 63 164–171 (2016).
14. Mariana Lima da Costa Valente, Denise Tornavoi de Castro, Ana Paula Macedo, Antonio Carlos Shimano, Andréa Cândido dos Reis: Comparative analysis of stress in a new proposal of dental implants. Materials Science and Engineering C 77 360–365 (2017).
15. Ingrassia, T., Nalbone, L., Nigrelli, V., Pisciotta, D., Ricotta, V.: Influence of the metaphysis positioning in a new reverse shoulder prosthesis. Springer International Publishing AG Advances on Mechanics, Design Engineering and Manufacturing, Lecture Notes in Mechanical Engineering, DOI 10.1007/978-3-319-45781-9_47 (2017).
16. Pankaj Dhatrak, Uddhav Shirsat, Sumanth S, Vijay Deshmukh: Finite Element Analysis and Experimental Investigations on Stress Distribution of Dental Implants around Implant-Bone Interface. Materials Today: Proceedings 5 5641–5648 (2018).
17. Ingrassia, T., Nigrelli, V., Ricotta, V., Nalbone, L., D'Arienzo, A., D'Arienzo, M., Porcellini, G., A new method to evaluate the influence of the glenosphere positioning on stability and range of motion of a reverse shoulder prosthesis, 2019, Injury, 50, pp. S12-S17

18. Rinaudo, A, D'Ancona, G, Lee, JJ, Pilato, G, Amaducci, A, Baglini, R et al. Predicting Outcome of Acute Aortic Dissection with Patent False Lumne by Computational Flow Analysis. Cardiovascular Engineering and Technology 5(2),176-88 (2014).
19. Mancuso, A., Pitarresi, G., Tumino, D., 2018, Using FEM simulation to predict structural performances of a sailing dinghy. International Journal on Interactive Design and Manufacturing (IJIDeM), 12(3), 811-822
20. NarendraKumar U, Arun Tom Mathew, Nikhil Iyer, Febin Rahman, I Manjubala: A 3D Finite Element Analysis Of Dental Implants With Varying Thread Angles. Materials Today: Proceedings 5 11900–11905 (2018).
21. D'Ancona, G, Amaducci, A, Rinaudo, A, Pasta, S, Follis, F, Pilato, M et al. Haemodynamic predictors of a penetrating atherosclerotic ulcer rupture using fluid–structure interaction analysis. Interactive CardioVascular and Thoracic Surgery 17(3),576-8 (2013)
22. Fragapane, S., Giallanza, A., Cannizzaro, L., Pasta, A., Marannano, G.: Experimental and numerical analysis of aluminum-aluminum bolted joints subject to an indentation process. International Journal of Fatigue, 80, 332-340 (2015)
23. Joel Ferreira Santiago Junior, Fellippo Ramos Verri, Daniel Augusto de Faria Almeida, Victor Eduardo de Souza Batista, Cleidiel Aparecido Araujo Lemos, Eduardo Piza Pellizzer: Finite element analysis on influence of implant surface treatments, connection and bone types. Materials Science and Engineering C 63 292–300 (2016).
24. Cirello, A., Cucinotta, F., Ingrassia, T., Nigrelli, V., Sfravara, F., Fluid–structure interaction of downwind sails: a new computational method, 2019, Journal of Marine Science and Technology (Japan), 24 (1), pp. 86-97.
25. Suchat Aumnakmanee, Nantakrit Yodpiji, Nattawut Jantong, Manutchanok Jongprasithporn: Finite element analysis of dental implant prosthetics. Materials Today: Proceedings 5 9525–9534 (2018).
26. Ingrassia, T., Lombardo, B., Nigrelli, V., Ricotta, V., Nalbone, L., D'Arienzo, A., D'Arienzo, M., Porcellini, G., Influence of sutures configuration on the strength of tendon-patch joints for rotator cuff tears treatment, 2019, Injury, 50, pp. S18-S23
27. Ingrassia, T., Nalbone, L., Nigrelli, V., Ricotta, V., Pisciotta, D., Biomechanical analysis of the humeral tray positioning in reverse shoulder arthroplasty design, 2018, International Journal on Interactive Design and Manufacturing, 12 (2), pp. 651-661.
28. Camarda, L., Pitarresi, G., Moscadini, S., Marannano, G., Sanfilippo, A., D'Arienzo, M.: Effect of suturing the femoral portion of a four-strand graft during an ACL reconstruction. Knee Surgery, Sports Traumatology, Arthroscopy 22(5), 1040-1046 (2014)
29. Petr Marcian, Jan Wolff, Ladislava Horackova, Jozef Kaiser, Tomas Zikmund, Libor Borak: Micro finite element analysis of dental implants under different loading conditions. Computers in Biology and Medicine 96 157–165 (2018).
30. Igor Linetskiy, Vladyslav Demenko, Larysa Linetska, Oleg Yefremov: Impact of annual bone loss and different bone quality on dental implant success – A finite element study. Computers in Biology and Medicine 91 318–325 (2017).