

# We are IntechOpen, the world's leading publisher of Open Access books Built by scientists, for scientists

6,900

Open access books available

186,000

International authors and editors

200M

Downloads

Our authors are among the

154

Countries delivered to

TOP 1%

most cited scientists

12.2%

Contributors from top 500 universities



WEB OF SCIENCE™

Selection of our books indexed in the Book Citation Index  
in Web of Science™ Core Collection (BKCI)

Interested in publishing with us?  
Contact [book.department@intechopen.com](mailto:book.department@intechopen.com)

Numbers displayed above are based on latest data collected.  
For more information visit [www.intechopen.com](http://www.intechopen.com)



---

# Finite Elements Method in Implant Prosthetics

---

Iulia Roateși

Additional information is available at the end of the chapter

<http://dx.doi.org/10.5772/64969>

---

## Abstract

This chapter is devoted to the study of behavior of functional loadings for implant prosthetics rehabilitation by finite elements method (FEM). It presents a numerical calculation of stress, displacement, and strain in implant and surrounding bone, which is used to assess risk factors from a biomechanical point. The masticatory forces are simulated by axial and/or non-axial loads, and they are responsible for the biomechanical response of the bone-tissue-implant-crown system. This chapter represents an analysis of this response in view of highlighting the factors involved in implant stability and success. The safety factor for different loading cases is calculated as well. A good agreement with other study results and clinical studies is obtained.

**Keywords:** finite elements methods, implant prosthodontics, biomechanics, stress distribution, numerical solutions

---

## 1. Introduction

This chapter presents a study of the biomechanical behavior of dental implant prosthodontics in a mandible under masticatory loads using numerical models made by the finite elements method (FEM). This method is widely used in the study of dental implants and has proven very useful in analyzing the distribution of strains and stress in the entire bone tissue, implant, and crown structure [1].

These studies are equally relevant in achieving optimal design of the implant, in osseointegration study, in oral rehabilitation study by developing various loading and orientation versions of the implant, and so on [2].

---

Numerical analysis of implants is useful for doctors in a deeper and thorough understanding of the characteristics of the state of stress and strain in the bone structure to ensure success of implants [3].

FEM is a numerical method developed by engineers and substantiated by mathematicians to find approximate solutions to complex structure problems. This method is also successfully used in medicine for biological structures and, in particular, in dentistry.

The method consists of dividing (mesh) the problem structure domain in smaller areas (finite element) connected by nodes, called mesh; finding the solution to these subdomains, finite elements (FEs), considering both loads (forces, pressures, etc.) applied to the structure boundary, as well as the boundary conditions and finally assembling these elementary solutions for finding global solutions.

FEM is considered by many specialists as one of the most common numerical methods for structural analysis. Regardless of their type and complexity, FEM is used in industry and science, taking increasingly greater scale in medicine and, certainly, its usefulness being quite rigorously outlined in dentistry and all of its specialties [4].

The large variety of types of finite elements, of loads that can be taken into consideration, of material modes of behavior (linear or non-linear), of the application regime (static, elastic, or dynamic), of the types of analysis (structural response, modal analysis, fatigue, fracture, optimization, etc.) gives the method and, therefore, the user the possibility to carry out an accurate calculation for any problem, with results as close as possible to reality [5].

FEM possibilities are closely linked to the performance of programs and computers. Currently, software products of analysis by FEM are made, which, under increasingly high hard performance, enables the possibility that the size or difficulty of the problem to be solved to no longer constitutes an insurmountable obstacle.

Correctly solving the structure analysis problems by FEM is based primarily on creating a numerical model as suitable as possible; this model should be based on the correct understanding of the problem to be solved and equally on the knowledge of the theoretical foundations of the method.

The great versatility and efficiency of FEM determined its utilization in areas as highly diverse as mechanical structures, fluids, thermal processes, and recently in areas of medicine such as orthopedics, stress flow, dental medicine, and so on.

Therefore, FEM is currently one of the most powerful tools for investigating many phenomena of the most complex and highly diverse areas.

The FEM uses as a starting point an integral model of the phenomenon under consideration [6, 7]. It applies separately for a series of small parts of a continuous structure obtained by the meshing process, known as finite elements, connected to each other at points called nodes [8]. These finite elements must be designed so that their ensemble reconstructs as closely as possible the actual analyzed structure. In principle, these connections must be designed as to allow a numerical convergence to the exact solution when the structure is discretized in finite elements with dimensions increasingly reduced.

### 1.1. Finite elements analysis steps

Finite elements analysis of a structure should take the following steps [6–8]:

#### 1. *Dividing analysis area in finite elements (mesh)*

Meshing a structure means that it is subdivided in a seldom number of finite elements or in a numerical integration point mesh, interconnected in their exterior nodes. During this operation, the types of finite elements to be used will be chosen and their distribution among the meshed area is established, resulting in their number, size, and form.

Meshing is done by computer. The finite element type is defined by several characteristics, such as dimension (one-, two-, three-dimensional), the number of the element's nodes, the associated approximation functions, and others. Choosing the finite element is of great importance for the necessary of internal memory, for the required calculation effort of the computer, and for results quality.

In case of two-dimensional structures, for modeling, triangular or quadrilateral finite elements can be used; the triangular elements ensure better opportunities in terms of contours geometry approximation, while quadrilateral elements accurately reproduce the distribution of stress. It is appropriate to use the elements as close as possible of the equilateral triangle or square. The use of very obtuse triangles or rectangle angles is not recommended, with too elongated elements. Similarly, in three-dimensional structures, tetrahedral or parallelepipedic types of finite elements can be used.

The starting point for the mathematical construction of various finite elements methods is represented by respecting the following principles:

- considering approximations based on the use of simple elements for which we have provided a solution;
- increasing accuracy of the calculation by refining the mesh.

### 1.2. Effect of meshing on the numerical results

Calculation results (displacements strain and stress), which are obtained by FEM, are dependent of the meshing choosing. For this reason, there are situations, especially in the case of complicated geometries when the problem addressed with these methods should be investigated in several meshing variants and subsequently sorting out the results [8].

On the other hand, the effect of errors increases with the number of elements used. Numerical errors are due to truncation, rounding, and of input data errors.

To study the influence of meshing, the most common method is to halve the mesh and compare it, and if the results are negligible, the analysis is considered acceptable. To effectively achieve more accurate results by successive refining of the finite elements mesh, the following criteria must be considered:

- Each previous mesh should be reflected in the new one;



- Each point of the structure must always be found in a finite element;
- The approximation function (type of element) should remain the same when passing from one element to another.

Nevertheless, it must be noted that from a certain number of finite elements, results can no longer be improved by increasing their number, and changing the type of finite element used becomes compulsory.

The model obtained by meshing the structure into finite elements must meet the following requirements:

- to represent with sufficient fidelity the actual behavior of the structure;
- to allow easy generation of results (displacements, strains, stress);
- to not involve an excessively high labor for input data preparation or results processing and hence a very long work time for the computer and a great part of its memory.

Some of these requirements are becoming less restrictive following the improvement of software and technical performance of computers.

It should be emphasized that computer programs are capable of performing automatic meshing, being able to perform even an automatic refinement of meshing in areas where there are large gradients of stresses (strains).

## *2. Establishment of finite element equations (elemental equations)*

Material behavior within a finite element is described by finite element equations called elemental equations. These form a system of equations of the element. Basic equations can be derived directly on a variational way, through, for example, residues method (Galerkin) or by the energy balance method.

## *3. Assembling the elemental equations in the equations system of the structure*

The whole structure behavior is modeled by assembling systems of finite elements equations in the system of structure equations, which physically means that the structure balance is conditioned by the balance of finite elements. The assembly requires that in the common nodes of the elements, the unknown function or functions to have the same value.

## *4. Implementation of boundary conditions and solving the system of the structure equations*

The equations system obtained from implementing appropriate boundary conditions suitable for the problem under consideration is solved by one of the common procedures, for example, by Gauss elimination or by Cholesky method, and so on, obtaining function values in nodes. These are called primary or first-order unknowns.

### *5. Performing additional calculations to determine the secondary unknowns*

In some problems, after finding the primary unknowns, the analysis closes. In other problems, however, knowing only the primary unknown is not sufficient, the analysis must proceed with the determination of the secondary unknowns. These are higher-order derivatives of the primary unknowns. Thus, for example, in dental implants analysis, primary unknowns are the nodal displacements. With their help, in this step, secondary unknowns are determined, represented by strains and stress [6, 8].

## **2. Material and method**

### **2.1. The use of FEM in dental implants studies**

FEM is very useful in the study of dental implants because it determines stress, strain, and displacement both in the implant, crown, as well as in the bone tissue. This calculation plays a crucial role both in the optimal design of implants [9] and in determining the factors that control the whole process of post-implant osseointegration [3].

In recent years, studies that are based on the use of FEM in oral implantology became very numerous and are dedicated either to oral rehabilitation or to analysis of the state of stress in and around the implant in the peri-implant bone (cortical and trabecular bone). We can mention works that use two-dimensional (2D) representations of the implant and mandible or maxilla and consider a geometry of the implant-bone system and axial-symmetric loads [10] or numerous three-dimensional (3D) studies [11], which model the bone geometry or loads application in a proper way. On the other hand, there are studies that consider a more realistic modeling of mastication, namely cyclical dynamic loads directed in an occlusal angle [12].

The importance of FEM numerical analysis in the study of dental implants involves several aspects. The method is equally useful both for clinicians, by investigating alternative treatments, and for dental implants producers [13]. They can change the macro-design according to clinical benefits. The purpose of improving the design and use of dental implants is represented by the bone absorption as reduced as possible in the region around the implant, certain micro-displacement of the blunt, a better distribution of the loading on the implant structures [14]. All these properties are often related to the biomechanical behavior and should be investigated not only in clinical trials but also in FEM studies [2].

A new design of dental implants and materials must be subjected to thorough investigation and compared to traditional structures. FEM analysis allows the comparison of old and new treatment modelings, taking into account the limitations and deeper understanding of the applicability areas [15].

FEM studies are not generally designed for clinicians without the help of engineers. Typically, clinicians have limited knowledge of modeling and simulation using computers and therefore they need the help of engineers. Therefore, collaboration between a clinician and an engineer in this field is very useful. The clinician should devote too much time to acquire adequate

knowledge on techniques of computational for the preparation and development of a numerical model implementation. However, the clinician must manage the study and provide necessary directions and indications concerning the dental implant parts, bone physiology, masticatory forces, the implant bonds with the bone, generally, the dental implantology matter. Also, the clinician should make every effort to maintain contact with the engineer to achieve effective assessment and adaptation of the numerical model [16].

The number of elements and nodes can be increased to achieve a more detailed modeling. But this can be time consuming, which could hinder the development of the model and complicate the computational calculations. Therefore, the engineer must clearly understand the purpose of the study, the boundaries, the limitations that can be applied to the model, and the number of elements can be increased only in the area of interest, therefore in the vicinity of the implant.

The difficulties in finite element modeling in dental implants study are linked on one hand to the assignment of material properties for biological materials: cortical bone, trabecular bone, gingiva; on modeling of loads that act on the bone-tissue-implant structure [17], and also on the actual faithful realization of the geometric model of the implant, which can be extremely laborious in the case of accurate modeling [11, 18].

As previously mentioned, 3D models are widely used. Given the nature of applied loads, unbalance of loads, and of geometric structures involved as well as their complexity, 2D models are unsatisfactory, 3D models being rather preferred [13, 19]. 2D models [20] cannot simulate the behavior of real structures as realistic as 3D models, so, the latest research focuses on 3D models [3]. The present paper continues the modern trends of recent years of using detailed 3D models that are much closer to reality both in geometric modeling accuracy loads application and in boundary conditions.

This is the case of the present study, which consists of the calculation and analysis of the state of stress and strain and displacement of the implant and its interaction with the mandibular bone.

In implants cases, the most critical area is that in which there is maximum stress concentration [9]. This is the neck of the implant and the surrounding area, namely the cervical edge (marginal bone) [18]. Therefore, this area should be kept as intact as possible in order to maintain a structural and functional bone-implant interface [21].

## **2.2. Numerical analysis program**

Modeling and simulation of dental implants behavior are made using Solid Works program regarding the geometric model development to simulate the implant and bone-tissue structure and Cosmos program for numerical solving of the obtained scheme [22].

Cosmos program is one of the best professional programs of FEM analysis of the continuum mechanics. This program was used in the version of its integration into the Solid Works product—one of the most powerful media of structure modeling—thus ensuring an efficient and accessible interface, both during data preprocessing and during the post-processing of

obtained results using graphs, isocurves, isosurfaces to represent the values of the studied entities fields, and so on.

Since implementing the geometry model of dental implant, crown, and bone tissue requires special preprocessing resources, SolidWorks software was used. The geometric model made with this program was exported and used for the Cosmos program calculations.

The present study analyzes the insertion of a dental implant in a section of the mandible, focusing primarily on highlighting and predicting areas of stress concentration both in the trabecular bone and in the cortical bone for different clinical situations and loads caused by mastication. These areas are the most vulnerable areas, where potential failures, fractures, or damage structure may occur.

### 2.3. The geometric model of the dental implant

#### 2.3.1. General considerations

The first step of finite element modeling is to provide implant-crown-bone-tissue structure by creating a geometric model. The next step is choosing materials for structural components [23, 24]. Their properties are available in SolidWorks materials library and studies based on experimental determinations [19, 25].

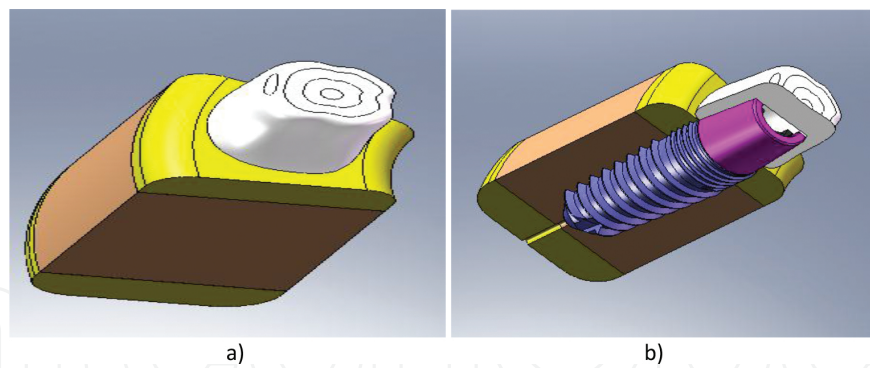
In our case, the analysis model is static and materials are considered elastic. We indicate the material constants for the different materials involved in the modeled structure, such as Young elasticity module ( $E$ ), Poisson coefficient ( $\nu$ ), the density, tensile strength, and yield strength. The third step is the application of restrictions related to boundary conditions and then applying the loads to simulate the mastication forces in the considered problem.

Both the geometric model and the complete finite elements model of the implant components and of the bone tissue were performed with great accuracy, considering the construction and functional details (radius of fillet, fillets, releases, and contact) for the results to be as close to reality as possible. All of this can be seen in **Figures 1–12**.

#### 2.3.2. Geometric model components

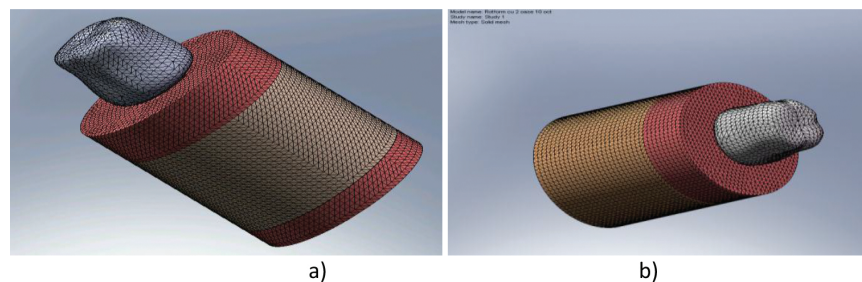
The case of a structure consisting of a 13.5-mm long implant, with a diameter of 3.8 mm with two threaded zones (fine step and big step) placed in a section of the mandible with extension of about 20 mm from the implant axis as shown in **Figures 1(a)** and **1(b)**, was considered [26].

The geometric model consists on one hand of the biological material, mandibular part, which comprises the cancellous bone, cortical bone, and gingiva tissue, and on the other hand of the implant, prosthetic blunt, and crown. The implant is usually made of titanium alloy and crown of ceramic [27]. In the studied Denti implant case, the blunt is made of magnesium alloy (**Figure 1b**).



**Figure 1.** Geometric model: (a) overview and (b) section.

The geometric model of the bone tissue may also be considered in a simpler geometrical shape, for example, in the form of a cylinder, which comprises a layer of trabecular bone lined both in the upper and in the lower sides by a thinner layer of cortical bone, as shown in **Figure 2(a)**. The geometric model of the bone tissue could be simplified by considering a geometric shape in the form of a cylinder, but which contains a layer of trabecular bone bounded only at the top by a thinner layer of cortical bone, as shown in **Figure 2(b)**, which represents bone constituents who contact the implant.



**Figure 2.** Geometric model, bone tissue, simplified bone versions, with the implant and the crown inserted: (a) trabecular bone placed between two cortical bone disks (marked in red); (b) trabecular bone superior bounded by a disk of cortical bone (marked in red).

For structures shown in **Figures 1(a,b)** and **2(a,b)**, various calculations by applying appropriate boundary conditions and some axial and lateral loads between 100 and 400 N have been carried out, which are simulating the masticatory forces [28].

Calculations revealed no significant differences either in values obtained for stress, strains, and displacements, or in their localization which is why it can be considered for subsequent calculations the simpler geometry of the bone tissue as shown in **Figure 2(a,b)**. As the time of creating the geometric model and the running time is much lower in simplified geometry, considering a simplified geometry model is useful for quick guidance calculations, which allow running many cases, with variation of the different factors taken into account, for example, different values, localizations, and positions of the loading forces (simulating masticatory forces), types of material with different values of the material constants, and so on.

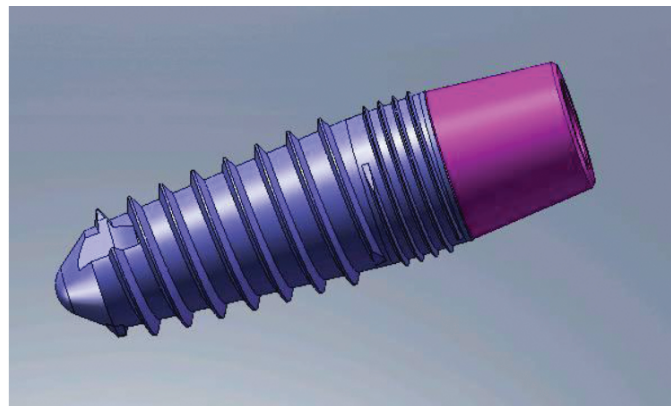


For problems that require highly accurate calculations in areas of interest, it is preferable to chose a geometry as close to reality as possible.

The analyzed structure components are implants, crown, and mandible portion. All these components were created by a computer and were used in the finite element calculation [26, 29].

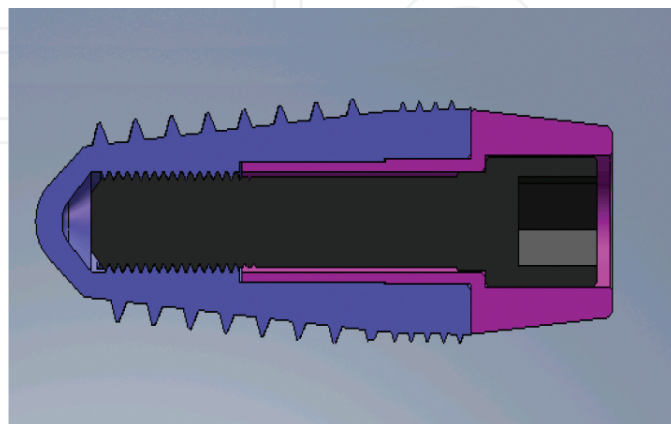
#### 2.3.2.1. *Implant*

It is cone-shaped with two threaded zones (see **Figure 3**).

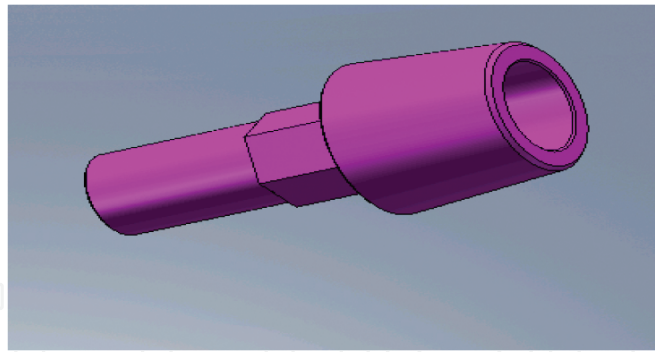


**Figure 3.** The geometric model of the implant.

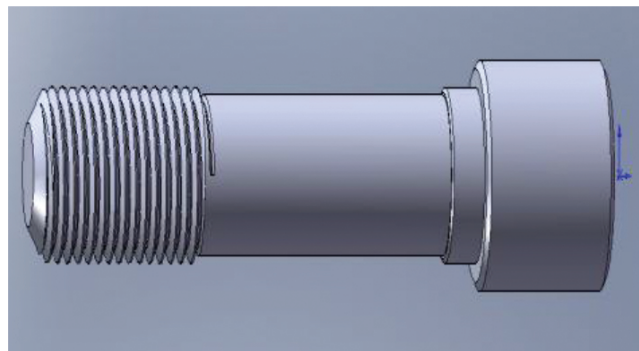
The interior of the implant (see **Figure 4**) allows installation without a threaded zone of the intermediate part (blunt) (see **Figure 5**) and at the top has a hexagonal bore for mounting with an Allen key in the mandible. The thread of the inner part is used to assemble the implant with the prosthetic blunt by a titanium screw M2 (see **Figure 6**).



**Figure 4.** Longitudinal section through the geometric model of the implant where implant system parts can be observed.



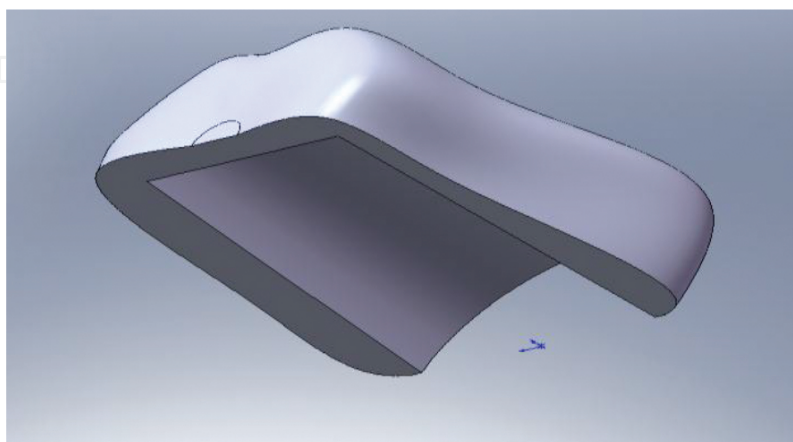
**Figure 5.** Geometric model of abutment.



**Figure 6.** Geometric model of screw.

#### 2.3.2.2. *Crown*

This component is made from ceramic and has been modeled by a geometric shape as close to the real one as possible (**Figure 7**).



**Figure 7.** Geometric model of crown in section.

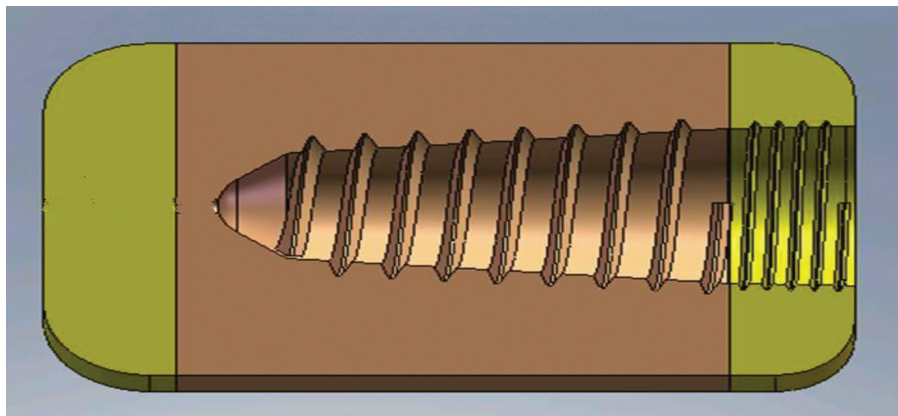


### 2.3.2.3. Bone tissue

This is the portion of the mandible around the implant at a distance from it which does not influence the stress and strain of the studied ensemble.

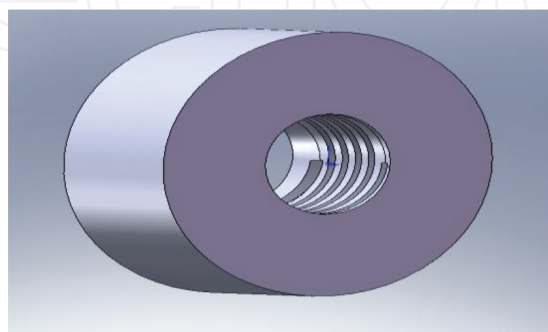
Adoption of the mandible portion size was based on the Saint-Venant's principle (the new distribution of forces produces appreciable differences in stress in the area of application, but remain with no effect or insignificant effect at large distances from the place of forces application) and by preliminary numerical simulations which confirmed the correct choice of size [30].

Modeling the mandible portion took into account the different structure of the bone portion (cortical and trabecular) by assigning suitable material properties in respective areas (see **Figure 8**) [17].

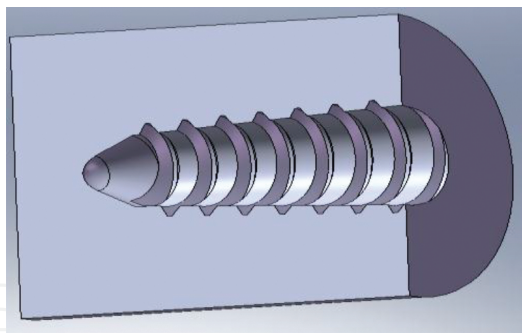


**Figure 8.** Geometric model of bone as a piece of mandible, in section: cancellous bone portion bordered by the upper and lower thinner layers of cortical bone.

For example, in the case of simplified modeling of the bone tissue in a cylindrical shape, the cortical bone is shown in **Figure 9** and the trabecular bone is shown in **Figure 10**.

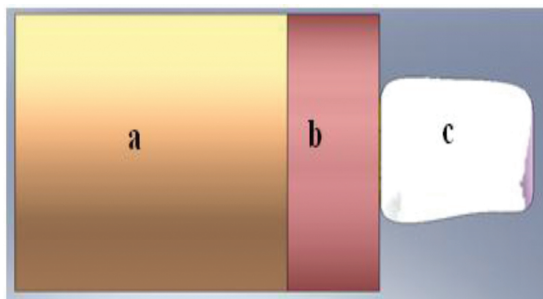


**Figure 9.** The geometric model of the cortical bone component in the simplified version in the form of a cylinder. The implant thread with a small step can be seen.

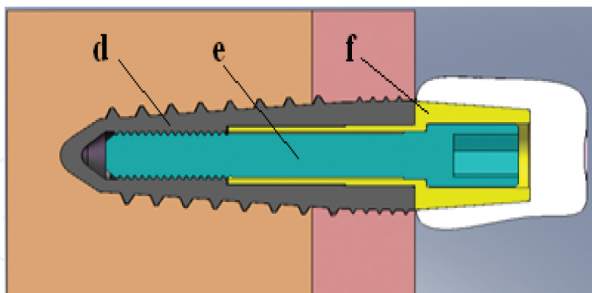


**Figure 10.** Geometric model of cancellous bone component in the form of a simplified cylinder version. The implant thread with a large step can be seen.

For the same example, the whole geometric model, consisting of the trabecular bone side, cortical bone, implant, and crown, is shown in **Figure 11**, and a complete section in **Figure 12**.



**Figure 11.** The geometric model of the assembly, the visible portion in the image is represented by compact bone (a), trabecular bone (b), and crown (c).



**Figure 12.** The longitudinal section of the system, composed of bone portions represented by compact bone, trabecular bone, the implant (d) fixed along the two portions of bone, the clamping screw (e), blunt (f), and crown.

All components were modeled respecting all elements and technical details (threads, taperings, releases, etc.).

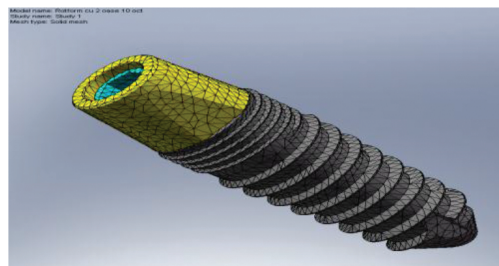
The assembly was designed to transmit the mastication force from the crown to blunt and then to the upper part of the implant; mastication force transmission mechanism does not intervene in any threaded part [31].

## 2.4. The dental implant finite elements model

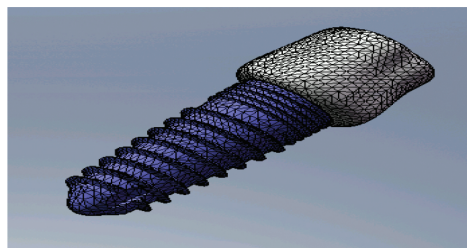
### 2.4.1. The finite elements

3D finite elements model used for the study of an implant in a mandible portion was built using SolidWorks software and uses tetrahedral elements both in the implant and in the bone. In the following, we represent one meshing option that easily allows viewing of the considered details.

The FE model is made of several parts corresponding to implant components, as shown in **Figures 13** (FE model of the implant) and **14** (FE model of implant-crown assembly) [26].

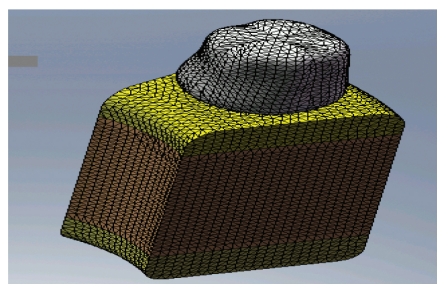


**Figure 13.** FE model of implant and abutment.

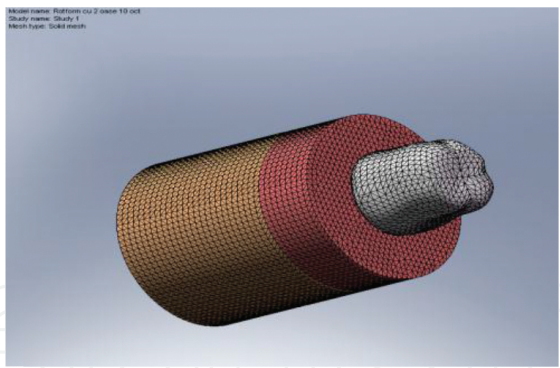


**Figure 14.** FE model of implant-crown assembly.

**Figures 15** and **16** represent a complete FE model of the implant and mandibular portion, respectively, complete FE model for the implant and mandibular cylindrical portion.



**Figure 15.** Complete FE model of crown, implant, and mandible portion.



**Figure 16.** Simplified FE model (cylindrical mandible portion, implant, crown).

Usually, when a problem is to be modeled, first, simple models are created, beginning with the geometry for a quick development, calculation, and running, considering different scenarios (e.g., loading size, localization of loading application, etc.). Further, more complicated models are created, in order to bring it as close as possible to reality. The main difference between the two models is the easy way to construct, to handle, to run for a much shorter time, and so on, the simpler one. But the complete model offers more accurate information, especially the localizations of critical zones, despite the difficulty of its construction and a longer running time.

The features of present numerical studies obtained on the two models, full (portion of mandible) and simplified (cylindrical), are presented in **Table 1**.

Information about the mesh	Full model	Simplified model
Element size	0.4 mm	0.4 mm
Tolerance	0.02 mm	0.02 mm
Number of elements	216904	165382
Number of nodes	318030	243874
Meshing time (hh; mm; ss):	00:07:44	00:01:18

**Table 1.** Mesh information.

The large number of finite elements relative to the size of the implant is explained by the existence of fine threads in the structural elements. Meshing fineness appeared as a need for a closer-to-reality modeling of constructive forms of great finesse, as threads, releases, and so on.

2.4.2. *Material models and their characteristics*

In this paragraph, the material types and the characteristics of the material are presented for each component, namely the bone tissue (composed of trabecular bone and cortical bone), implant, and crown.

No.	Component name	Material	Weight	Amount
1	Crown	Ceramics	0.000223658 kg	$9.72426 \times 10^{-008} \text{ m}^3$
2	Prosthetic blunt	Magnesium alloy	$8.17675 \times 10^{-005} \text{ kg}$	$4.08837 \times 10^{-008} \text{ m}^3$
3	Trabecular bone	Trabecular bone	0.00162148 kg	$1.08099 \times 10^{-006} \text{ m}^3$
4	Cortical bone	Mandibular cortical bone	0.000677504 kg	$3.38752 \times 10^{-007} \text{ m}^3$
5	Implant $3.8 \times 11.5$	Titanium alloy Al- 4VS Ti6	0.000436557 kg	$9.85725 \times 10^{-008} \text{ m}^3$
6	Screw	Titanium alloy Al- 4VS Ti6	0.000172112 kg	$3.88621 \times 10^{-008} \text{ m}^3$

**Table 2.** Data on the used materials.

Ceramics (crown)		
Constant name	Value	Units
Elastic module	$2.2059 \times 10^{+011}$	N/m <sup>2</sup>
Poisson coefficient	0.22	
Shear modulus	$9.0407 \times 10^{+010}$	N/m <sup>2</sup>
Density	2300	kg/m <sup>3</sup>
Traction resistance	$1.7234 \times 10^{+008}$	N/m <sup>2</sup>
Yield strength	$5.5149 \times 10^{+008}$	N/m <sup>2</sup>
Thermal expansion coefficient	$1.08 \times 10^{-005}$	/Kelvin
Thermal conductivity	1.4949	W/(m.K)
Specific heat	877.96	J/(kg. K)

**Table 3.** Material constants used for ceramics.

Magnesium alloy (prosthetic abutment)		
Constant name	Value	Units
Elastic module	$4.2 \times 10^{+010}$	N/m <sup>2</sup>
Poisson coefficient	0.33	
Shear modulus	$7.7 \times 10^{+010}$	N/m <sup>2</sup>
Density	2000	kg/m <sup>3</sup>
Tensile strength	$4.2 \times 10^{+008}$	N/m <sup>2</sup>
Yield strength	$1 \times 10^{+008}$	N/m <sup>2</sup>
Thermal expansion coefficient	$1.5 \times 10^{-005}$	/Kelvin
Thermal conductivity	24	W/(m.K)
Specific heat	590	J/(kg. K)

**Table 4.** Material constants used for magnesium alloy.

These data are provided in [1, 32, 33] and by the technical presentation of implants used in the calculation. They were used as input to perform the numerical calculations.

For an easier presentation, data are summarized in tables as follows: type of material for each component, mass, and volume are shown in **Table 2**, and in **Tables 3–7** are represented characteristics of material that is used, respectively, for crown, for intermediate piece of implant, implant, trabecular bone, and cortical bone.

Titanium alloy (implant and the screw)		
Constant name	Value	Units
Elastic module	$1.048 \times 10^{+011}$	N/m <sup>2</sup>
Poisson coefficient	0.31	
Shear modulus	$4.1024 \times 10^{+010}$	N/m <sup>2</sup>
Density	4428.8	kg/m <sup>3</sup>
Tensile strength	$8.2737 \times 10^{+008}$	N/m <sup>2</sup>
Yield strength	$1.05 \times 10^{+009}$	N/m <sup>2</sup>
Thermal expansion coefficient	$9 \times 10^{-006}$	/Kelvin
Thermal conductivity	6.7	W/(m.K)
Specific heat	586.04	J/(kg.K)
Hardening factor (0.0–1.0; 0.0=isotropic; 1.0=kinematic)	0.85	

**Table 5.** Material constants used for titanium alloy.

Trabecular bone (mandible portion)		
Constant name	Value	Units
Elastic module	$1.8 \times 10^{+008}$	N/m <sup>2</sup>
Poisson coefficient	0.3	
Density	1500	kg/m <sup>3</sup>
Tensile strength	$2 \times 10^{+007}$	N/m <sup>2</sup>
Yield strength	$1.8 \times 10^{+007}$	N/m <sup>2</sup>

**Table 6.** Material constants used for trabecular bone.

Cortical bone (mandible portion)		
Constant name	Value	Units
Elastic module	$1.8 \times 10^{+010}$	N/m <sup>2</sup>
Poisson Coefficient	0.25	
Density	2000	kg/m <sup>3</sup>
Tensile strength	$1.5 \times 10^{+008}$	N/m <sup>2</sup>
Yield strength	$1.3 \times 10^{+008}$	N/m <sup>2</sup>

**Table 7.** Material constants used for cortical bone (mandible portion).



### 2.4.3. Contact modeling

In this analysis, we considered a model as close to real conditions as possible. In this regard, we paid special attention to the modeling of contact.

Contact zones are the threaded portions of the implant, mandibular segment, and screw. Contact phenomenon in these areas is modeled by special finite elements—contact elements, according to the actual behavior [33].

The contact model was used between adjacent surfaces of the threaded zones without allowing them for interpenetrating. In this way, up to seven pairs of contact surfaces were defined.

This contact modeling corresponds to a complete osseointegration, which corresponds to a period of implant analysis of more than 6 months after its insertion.

### 2.5. Boundary conditions

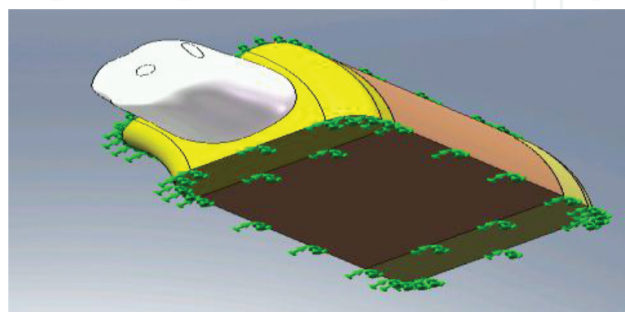
In general, structural analysis, boundary conditions are set in displacements and/or forces in those regions where these entities of the structure are considered to be known [6, 8].

These regions are considered restricted to remain fixed (if they have null displacements and/or rotations) during the numerical simulation or they may have non-zero values for specified displacements and/or rotations.

Non-zero displacement restrictions should be placed on boundaries of model for maintaining the balance of the solution.

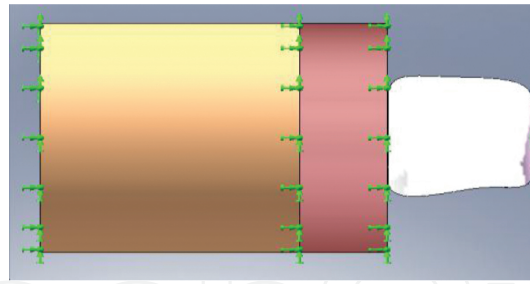
Also, restrictions should be placed in nodes that are away from the region of interest, in our case, in the vicinity of the implant. This is in order to prevent overlapping of the stress or strain field associated with the reaction forces to the bone-implant interface.

The models created with FEM presented in this study (**Figure 17**, complete model; **Figure 18**, cylindrical section of a simplified model), the lateral faces of bone tissue and the lower ones, were considered without moving, namely fixed (displacements, nodes are blocked on those faces in all directions).



**Figure 17.** Boundary conditions for FE model (green arrows).





**Figure 18.** Boundary conditions for the simplified FE model (green arrows).

## 2.6. Load application

Loosing marginal bone in the peri-implantation region may be a result of excessive occlusal forces [34]. To determine and understand this correlation, intensive research, including engineering principles, biomechanical relationships of living tissues, and mechanical properties of bone around the implant, is necessary.

In this context, the loads setting in an FE model is an important part of the study. Each component of the model is contributing to the final analysis after loading. In other words, from beginning to end, all FE analysis steps contribute to the ability to extrapolate masticatory forces in the area around the implant and prosthesis.

Masticatory forces can be forces of compression, traction, bending, and shear. Compressive forces try to push materials into each other, the traction forces pull them apart, separate entities and shear forces cause slides. Most detrimental forces that can increase stress around the bone-implant interface and prosthesis are traction and shear. These forces tend to damage the integrity of the material and cause stress concentration [2, 34].

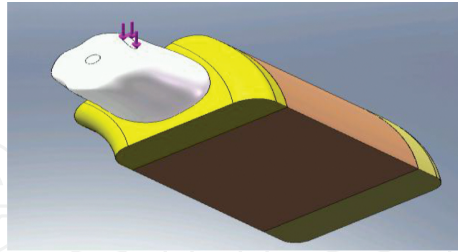
In general, the prosthesis-implant ensemble adapts to compressive forces. Under effective mastication, repetitive model of cyclical forces transmitted via dental implant causes load of the peri-implant bone. This generates a stress around the ridge and prosthesis. However, random cyclic forces of mastication are not easily simulated. Therefore, most FE studies use axial and/or non-axial forces [35].

Non-axial loadings generate a distinctive stress particularly in the cortical bone ridge. So, for a realistic simulation it is necessary to use a combination of vertical and/or oblique (axial and non-axial) forces. As mentioned, oblique loads are more destructive for the peri-implant bone region and clinically disruptive for the prosthesis.

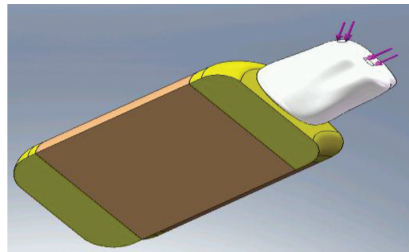
Masticatory force size can be variable depending on age, sex, and para-functional habits, and can vary from anterior to posterior [36]. Loadings simulating mastication forces generate stress concentrations that must be evaluated.

The loadings applied in the present study were provided by own experimental determinations (maximum values of masticatory forces women/men) and by information analysis from other studies [37–39]. In this study, we used both axial and non-axial loads. More precisely, more

sets of static loads were considered, namely axial loads, non-axial (**Figure 19**); axial and non-axial (**Figure 20**), applied to the crown.



**Figure 19.** Non-axial load application (arrows indicate the application direction).



**Figure 20.** Axial and non-axial load application (arrows indicate the application direction).

### 3. Results of FEM analysis of some clinical situations of implant prosthodontics rehabilitations

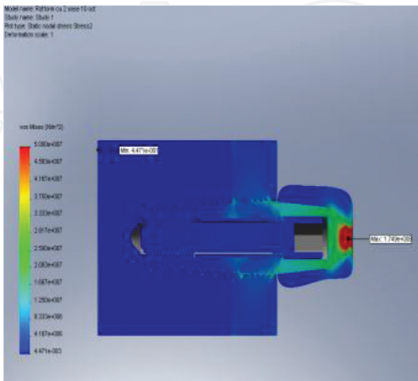
#### 3.1. Analysis of states of stress, strain, and displacement in the implant and in the bone tissue

Under the action of mastication forces, stress concentration increases in the prosthesis and bone. The stress is a representation of internal forces which are acting within a deformable body; it is a measure of the average force per unit of surface within the internal body where forces act. These internal forces occur in response to the application of external loadings on the considered body. Internal resistance after applying external loads cannot be measured. Therefore, a simple method is to measure the forces applied to the cross section.

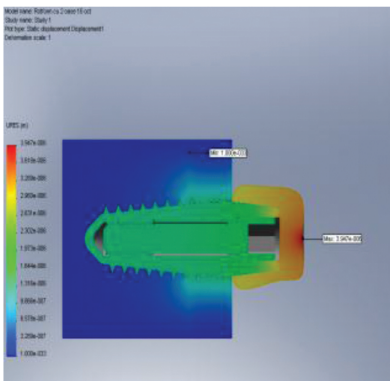
The FEM analysis of dental implants usually calculated von Mises stress (equivalent stress), a scalar quantity (number) characterizing stress size. This is very important in formulating the criteria of damage, plasticity, strength, and so on. This analysis is used to assess the effect of load forces on the region in the vicinity of dental implant or crown [34].

There is the convention that specifies that positive values of stress mean traction stress, while negative values of stress mean compressive stress.

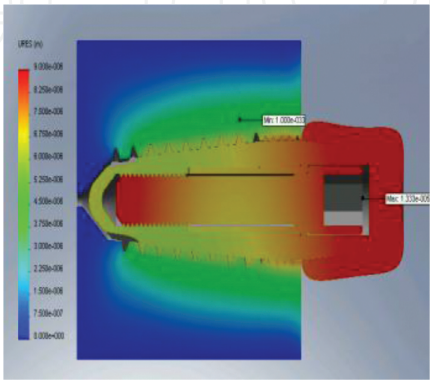
FEM numerical calculation results are shown as diagrams in which different colors signify areas of physical quantities considered of equal value, the minimum value areas to areas with maximum values. In **Figures 21–23**, the red-colored areas represent critical areas and blue-colored areas represent areas with the least loading.



**Figure 21.** von Mises stress field in the longitudinal plane section for axial force.



**Figure 22.** Displacement field in the longitudinal plane section for axial force.



**Figure 23.** Equivalent strain field in the longitudinal plane section for axial force.

In our analysis, loads that simulate axial and non-axial type masticatory forces were considered, with a range between 20 and 140 N.

In **Figures 21–23**, the most favorable areas are those with minimum values of stress, strain, and displacement, while areas with the greatest damage, high risk are characterized by higher values.

We will present the most important results regarding the distribution of the following entities of the study: von Mises stress field, displacement field, strain field, the safety factor (SF) for *axial loadings, non-axial loadings, and axial and non-axial loading simultaneously applied*.

### 3.1.1. Axial loadings

When axial loadings are applied, the results concerning the stress field distribution for the entire structure are shown in **Figure 21**, the displacement field in **Figure 22**, and the strain field in **Figure 23**.

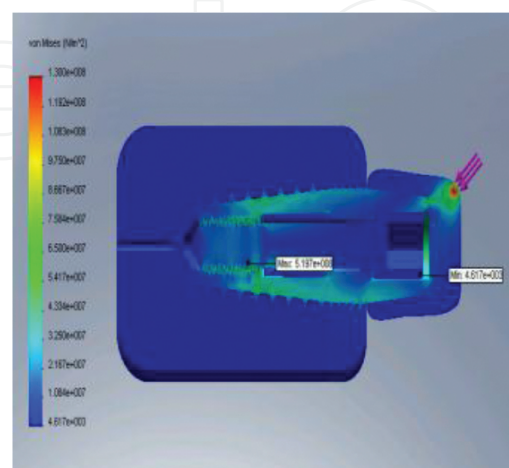
For example, **Figure 21** confirms the correctness of dimensions adopted for mandibular bone by the fact that stress in marginal areas of it is practically null.

**Figures 21–23** show fields symmetry due to the application of axial loads and they were obtained for a load value of 100 N.

We note that the case of axial loads is an ideal case, in reality this case is combined with non-axial loads.

### 3.1.2. Non-axial loadings

For non-axial loadings, we present the results for the von Mises stress field (**Figure 24**), displacement field (**Figure 25**), and strain field (**Figure 26**).



**Figure 24.** Von Mises stress field in longitudinal plane section for non-axial loadings.

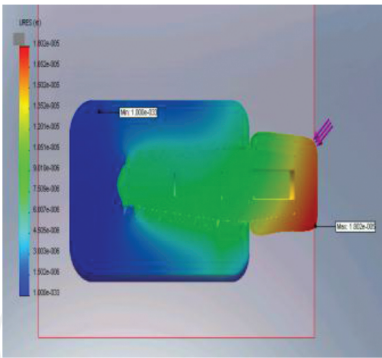


Figure 25. Displacement field in longitudinal plane section.

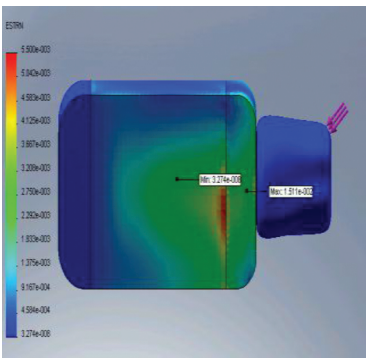


Figure 26. Strain field in longitudinal plane section for non-axial loadings for non-axial loadings.

Figures 24–26 confirm instead the asymmetry for stress, strain, and displacement fields due to the application of non-axial forces. Results from the figures presented here are obtained for a non-axial load of 120 N and represent the distribution of von Mises field stress, displacements, and strains of the entire bone-implant-crown system.

The distribution of these quantities—stress, displacements, and strains—is more suggestive if we consider their distribution separately only for bone tissue or for implant.

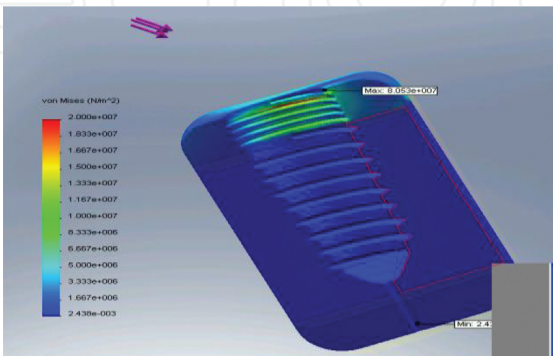
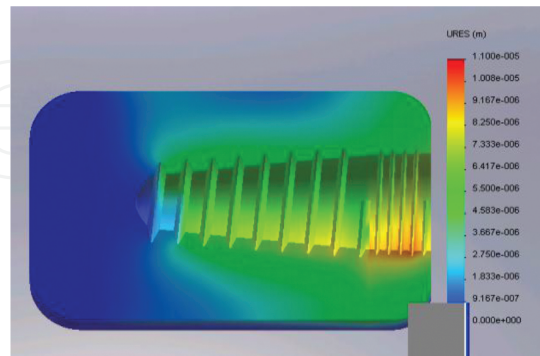
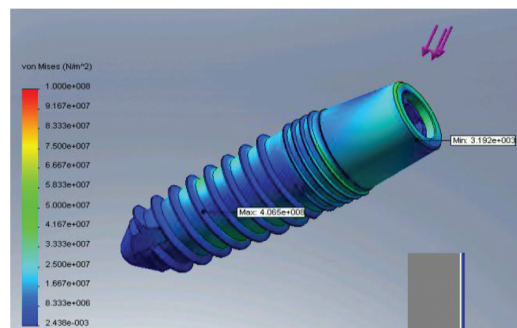


Figure 27. Von Mises stress in the bone tissue.

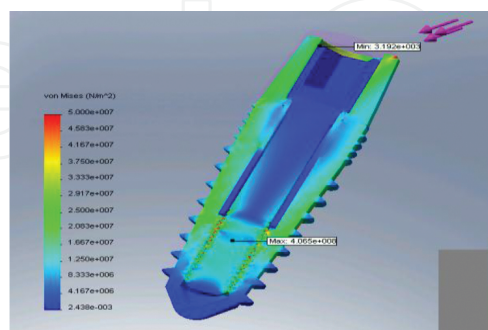
Thus, we present the results for the *bone tissue*: the von Mises stress distribution in **Figure 27**, displacements distribution in **Figure 28**; for *implant*: the von Mises stress distribution in **Figure 29** and von Mises stress distribution of the implant in the longitudinal section in **Figure 30**.



**Figure 28.** Displacements in the bone tissue.



**Figure 29.** Von Mises stress in the implant.



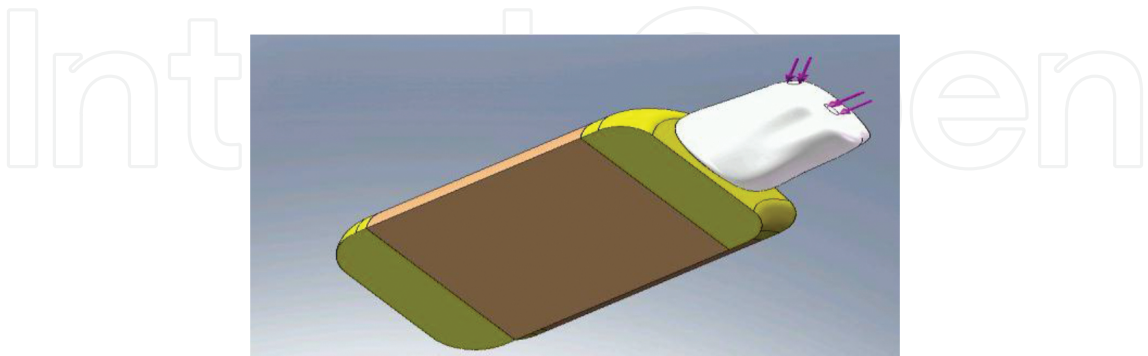
**Figure 30.** Von Mises stress in the longitudinal plane section of the implant.

From the results shown in **Figures 27** and **28**, it is observed that to obtain large displacements and stresses, which may become critical in the upper cortical bone of the implant neck, it is necessary to apply non-axial forces in the opposite direction.



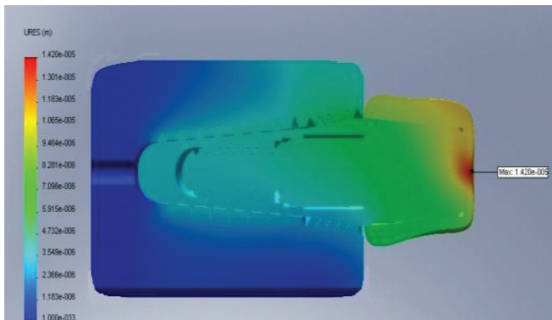
3.1.3. Axial and non-axial loadings applied simultaneously

A more realistic simulation of masticatory forces can be achieved by the simultaneous applications of axial (120 N) and non-axial loadings (20 N) (**Figure 31**). This case of application of forces is closer to the actual situation [26].

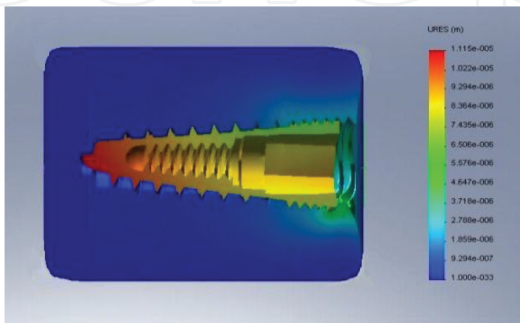


**Figure 31.** The simultaneous application of axial and non-axial forces.

In this case, for instance, the displacement distribution for the whole bone-implant-crown system is presented in **Figure 32**, while **Figure 33** shows the distribution of displacements only in the bone tissue.



**Figure 32.** Displacements in the case of application of axial and non-axial loads in the whole system.



**Figure 33.** Displacements in the case of application of axial and non-axial loads in bone tissue.



The comparison of these results with those obtained in the previous paragraph is very useful, as they allow an analysis concerning the localizations on which the forces are acting on. It appears that, for example, the maximum von Mises stress if only non-axial loads in Section 3.1.2 is  $5.197 \times 10^8$  Pa, unlike the calculations in this paragraph, the maximum value of von Mises stress is  $1.288 \times 10^8$  Pa. This fact is due to modeling of the masticatory forces which are applied over a larger area, case in which there is lower concentration (case of Section 3.1.3). If the application forces surface is smaller, then the maximum stress concentration is higher (case of Section 3.1.2).

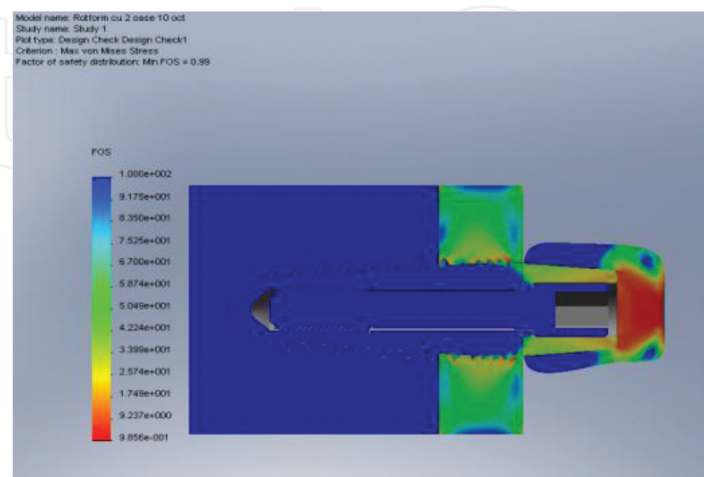
The stress is directly proportional to the force and inversely proportional to the surface area on which the force is applied. It is important to determine the surface area on which the loads are applied, as that fact influences considerably the calculation of maximum stress magnitude. For example, the area of the occlusal surface on which the rehabilitation is carried out is less than 4 mm, so that the amount of stress in much rehabilitation is in the range of MPa [40].

### 3.2. Safety factor determination

The CosmosWorks program has a section destined for determining the distribution of the factor of safety (FOS) or safety factor as a ratio between allowable stress limit values and actual stress values obtained by FEM calculation [22]. If  $SF < 1$ , the stress level is higher than the material limit and the structure is likely to fail [22].

Images are suggestive and may indicate the need for constructive appropriate solutions to eliminate dangerous areas that have a low safety factor.

**Figures 34** and **35** present safety factor distributions for implant determined by the von Mises method and Tresca, respectively, in the case of axial loading, and in **Figures 36** and **37**, the safety factor distributions for implant, determined by the von Mises and Tresca method in the case of non-axial loads, respectively.



**Figure 34.** Safety factor distributions in the longitudinal section, von Mises criterion, axial load.

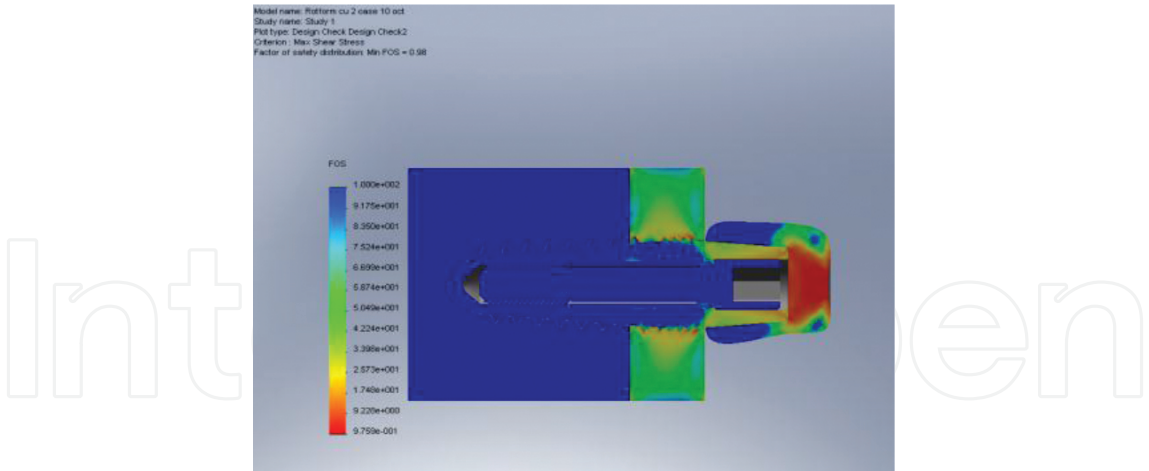


Figure 35. Safety factor distributions in the longitudinal section, Tresca criterion, axial load.

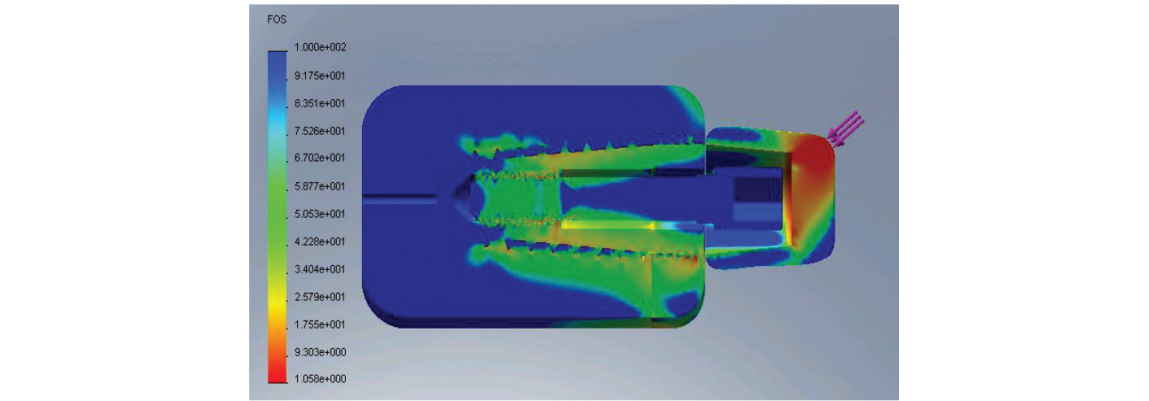


Figure 36. SF distribution in longitudinal section, the von Mises criterion, non-axial loads.

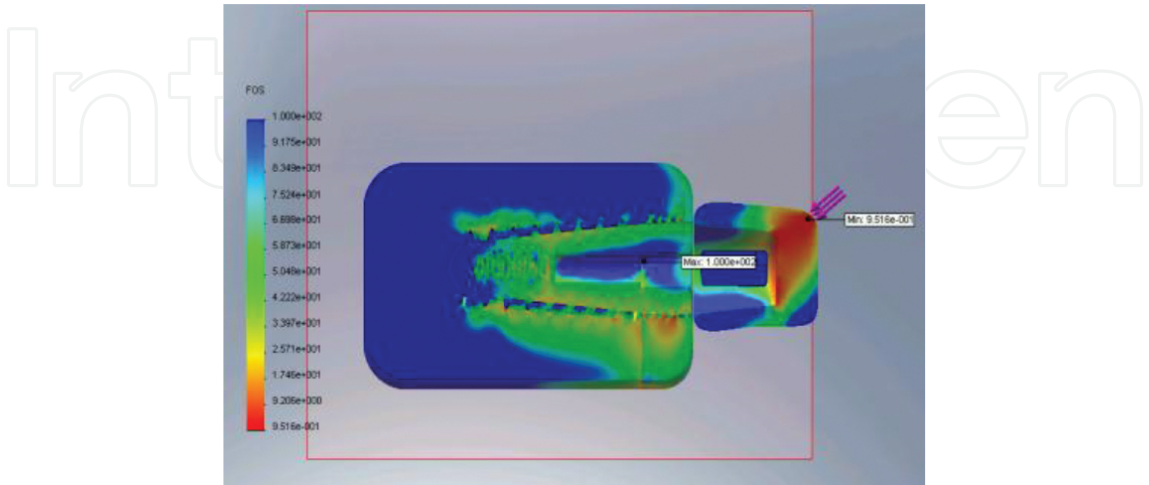
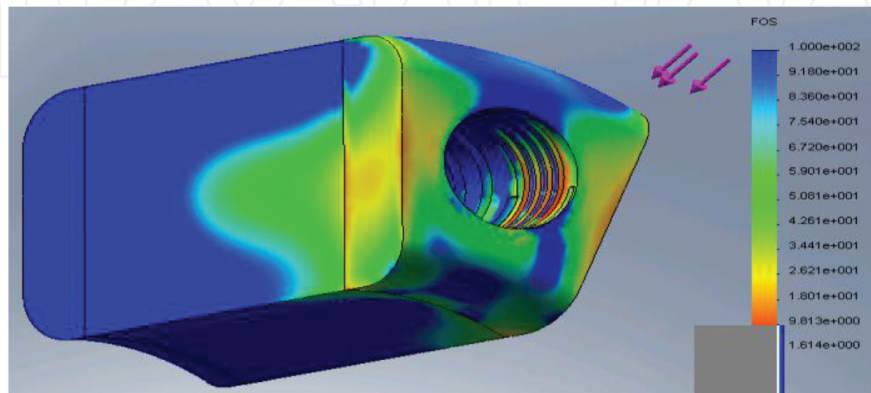


Figure 37. SF distribution in longitudinal section, the criterion Tresca, non-axial loads.

From these figures, the obtained results show that the most strained zone is the contact area of the crown and fine thread parts from the cortical bone area (red-colored areas in **Figures 34** and **35**). Between the two alternative calculations, there are small differences, both in terms of factor value and in terms of its distribution.

A suggestive description of the SF distribution can be achieved by considering the bone tissue only (**Figure 38**).



**Figure 38.** SF distribution in the bone tissue, Tresca criteria for non-axial loads.

From the obtained results of **Figures 36, 37** and **38**, asymmetry is observed due to the non-axial forces application, and also an area with very low safety factor in cervical bone area adjacent to the implant neck, on opposite application of oblique force.

## 4. Discussions

3D FEM analysis is very efficient for assessing the biomechanical behavior of a structure made of bone, implant, and crown under various loading conditions. In the past four decades, numerous studies have shown that FEM applied in dentistry is a very successful method used to investigate critical issues related to stress distribution [41]. Using detailed 3D models may be extremely useful in understanding the critical issues related to choosing the rehabilitation and application of procedures.

Results of a FEM analysis cannot be implemented directly in clinical situations, but it can design a model so as to simulate a real situation as well as possible [42]. Some limitations of the study using FEM are, however, the adoption of simplifications and assumptions.

FEM analysis should be interpreted with care. In most cases, the numerical studies of oral implantology use isotropic material, and not orthotropic, or anisotropic, as would be plausible [43].

On the other hand, the model by FEM is a static situation at a moment of loading application and not an actual clinical situation. In reality, structure loading is rather dynamic and cyclical.

Materials used in various branches of dentistry are supposed isotropic, homogeneous, and elastic, and so remain after loadings, which do not reflect the real situation [43, 44]. For example, periodontal ligament has non-linear mechanical properties and bone tissue is heterogeneous. The numerical results obtained should be more precise and rigorous if the material would be considered anisotropic and inhomogeneous, but they would lead to complex mathematical calculations on one hand, but more difficult, it would require more complicated laboratory experiments to determine material constants, especially for materials of living tissue type [45].

The results obtained by FEM use von Mises plasticity criterion, which is used in engineering rather for ductile material, for which the compression stress is equal to the traction stress as steel and aluminum. By contrast, brittle materials such as ceramics, cements, or composites resins had a compressive strength value significantly greater than the tensile strength [7].

The structure response is different under the action of asymmetric loads. When the structure is loaded compressively, no significant displacements occur due to higher compressive strength. The situation is different when loading is asymmetrical and when traction stress occurs. When a further lateral load is applied, traction stress is generated in areas with higher values than when vertical loads are applied in the same areas [43].

An increase in load does not cause a change in the stress distribution, only an increase in values. A loading that a structure is subjected can cause micro-cracks in certain parts of the structure, but not an immediate rupture [36].

Most dental breaks of used materials are caused by traction stress. A prevention of this phenomenon can be avoided by adjusting the occlusal surfaces. Masticatory forces vary between 11 and 150 N, while the maximum value of force is 200 N anterior, 350 N posterior, and 1000 N in bruxism [46, 47].

New developments in computer technology and modeling techniques make FEM a reliable and accurate approach in dealing with biomechanical applications.

In clinical practice, it may be considered that as these applications are being made by computer, assumptions and critical limitations can clearly affect the application of these results on a real scenario. Another aspect of the analysis by FEM is overemphasis results, due to the simplification applied to simulation models.

Although usually an advanced computing technology is used to obtain numerical results, there are many factors that affect clinical features such as macro- and micro-design, material properties, loading conditions, and boundary conditions.

Consequently, correlating the obtained FEM results with preclinical and clinical studies of long duration can help to validate results obtained by numerical simulations.

Note that several versions of meshing were created, to a fineness of it that does not lead to errors in the solution 3.5% higher than similar studies conducted by other authors [3].

Loading simulations must be as realistic as possible, containing simultaneous axial forces and non-axial [26, 29].

Results from our FEM analysis are consistent with data from the literature for similar models [48, 49]. Regarding optimizing dental implants, the following can be mentioned [50, 51]:

- Placing the implant should be done with caution so this would be as much as possible in the cortical bone area, as strain and stress values in the trabecular bone zone is very small, they may cause atrophy in the surrounding area.
- The neck of the implant should be long enough so that it departs from the soft tissue and any implant adherence has no effect on the mucous membrane. Any inflammation of the soft tissue and/or marginal bone resorption can jeopardize the stability of the implant.
- The neck of the implants should be quite robust, since the maximum stress concentrations occur in the neck of the implant. If the implant is not quite strong in this region, it can affect the integrity of the implant.
- There is a high risk of overload in the mesial and distal areas. This should be considered in patients where there is a narrow ridge of bone.
- Since loading does not necessarily mean it has overcome bone resistance, continuous loading more likely causes fatigue damage (micro-cracks in bone, marginal bone resorption). From a mechanical point of view, the presence of bone defects seems disadvantageous due to lack of bone support. Instead, the peri-implant bone stresses and strains are not only depending on the in vivo loading but also determined by bone quality (mechanical properties of bone) and amount (thickness of the cortical bone, density of the trabecular bone), periodontal state, oral hygiene, and other factors that may play an important role in marginal bone remodeling.

## 5. Conclusions

The present study constitutes a FEM calculation of stress, displacement, and strain in implant and surrounding bone, which is used to assess risk factors from a biomechanical point.

FEM studies have certain advantages over clinical studies, preclinical, and in vitro. First, patients are in no way affected by the application of new materials and new treatment that have not been previously tested. In the biomedical field, FEM is an important tool because it avoids the need for a traditional specimen, instead using a mathematical model that eliminates the need for a large number of teeth. FEM analysis helps in preparing the design, indicates the right materials, or a combination therefore to be used in various load conditions to reduce material consumption and/or failure of the clinical practice.

FEM analyses are useful for clinicians, although they require additional time to work, but they are a useful tool in predicting the implant choosing: type, shape, size. We believe that in the future, it will develop further, being available and accessible to a large number of doctors.

FEM simulations can be extended in numerous directions, such as parametric studies for the key factors involved in the analysis of the success/failure of the dental implant treatments; the analysis of the implant prostheses: bridges on the implants, connected crowns, and individual



crowns on the implants, in order to formulate exclusion criteria regarding the success of one of the therapeutic alternatives; comparative analysis of the various models of implants in terms of their stability and highlighting the factors involved in it.

## Acknowledgements

I would like to express my gratitude to Prof. Dr. E. Avram for his support for the geometrical model development and the FEM calculation. I also like to thank Prof. Dr. V. Năstăsescu for his high professionalism and expertise on FEM use.

## Author details

Iulia Roateși

Address all correspondence to: iulia.roatesi@gmail.com

Faculty of Dental Medicine, Titu Maiorescu University, Bucharest, Romania

## References

- [1] Gultekin B.A., Gultekin P., Yalcin S., Application of finite element analysis in implant dentistry. *Finite Element Analysis New Trends and Developments*. Rijeka, Croatia: Intech, 2012: pp. 21–54.
- [2] Vanegas-Acosta J.C., Landinez P.N., Garzon-Alvarado D.A., Casale R.M., *A finite element method approach for the mechanobiological modeling of the osseointegration of a dental implant*. *Comput Methods Programs Biomed*, 2011. 101(3): pp. 297–314.
- [3] Zheng L., Yang J., Hu X., Luo J., *Three dimensional finite element analysis of a novel osteointegrated dental implant designed to reduce stress peak of cortical bone*. *Acta Bioeng Biomech*, 2014. 16(3): pp. 21–8.
- [4] Brenner S.C., Scott R., *The mathematical theory of finite element methods*. Vol. 15. 2008: Springer.
- [5] Aversa R., Apicella D., Perillo L., Sorrentino R., Zarone F., Ferrari M., Apicella A., *Non-linear elastic three-dimensional finite element analysis on the effect of endocrown material rigidity on alveolar bone remodeling process*. *Dental Materials*, 2009. 25(5): pp. 678–90.
- [6] Avram E., Năstăsescu V., *Finite element method in fluid mechanics*. 1995: Military Technical Academy Press.

- [7] Huebner K.H., Dewhurst D.L., Smith D.E., Byrom T.G., *The finite element method for engineers*. 2008: John Wiley & Sons.
- [8] Zienkiewicz O., Taylor R., *The finite element method for solid and structural mechanics*. 2005: Butterworth-Heinemann.
- [9] Bahuguna R., Anand B., Kumar D., Aeran H., Anand V., Gulati M., *Evaluation of stress models in bone around dental implant for different blunt angulations under axial and oblique loading: A finite element analysis*. Natl J Maxillofac Surg, 2013. 4(1): pp. 46–51.
- [10] Bijjargi S., Chowdhary R., *Stress dissipation in the bone through various crown materials of dental implant restoration: a 2-D finite element analysis*. J Investig Clin Dent, 2013. 4(3): pp. 172–7.
- [11] Siadat H., Hashemzadeh S., Geramy A., Bassir S.H., Alikhasi M., *The effect of offset implant placement on the stress distribution around a dental implant: a three-dimensional finite element analysis*. J Oral Implantol, 2015. 41(6): p.646–651.
- [12] Ogawa T., Ogawa M., Koyano K., *Different responses of masticatory displacements after alteration of occlusal guidance related to individual displacement model*. J Oral Rehab, 2001. 28(9): pp. 830–41.
- [13] Chang C.L., Chen C.S., Yeung T.C., Hsu M.L., *Biomechanical effect of a zirconia dental implant-crown system: a three-dimensional finite element analysis*. Int J Oral Maxillofac Implants, 2012. 27(4): pp. 49–57.
- [14] Vivan Cardoso M., Vandamme K., Chaudhari A., De Rycker J., Van Meerbeek B., Naert I., Duyck J., *Dental implant macro-design features can impact the dynamics of osseointegration*. Clin Implant Dent Relat Res, 2015. 17(4):639–645.
- [15] Kasai K., Takayama Y., Yokoyama A., *Distribution of occlusal forces during occlusal adjustment of dental implant prostheses: a nonlinear finite element analysis considering the capacity for displacement of opposing teeth and implants*. Int J Oral Maxillofac Implants, 2012. 27(2): pp. 329–35.
- [16] Tang C.B., Liul S.Y., Zhou G.X., Yu J.H., Zhang G.D., Bao Y.D., Wang Q.J., *Nonlinear finite element analysis of three implant-blunt interface designs*. Int J Oral Sci, 2012. 4(2): pp. 101–8.
- [17] Marcian P., Borak L., Valasek J., Kaiser J., Florian Z., Wolff J., *Finite element analysis of dental implant loading on atrophic and non-atrophic cancellous and cortical mandibular bone –a feasibility study*. J Biomech, 2014. 47(16): pp. 3830–6.
- [18] Lee J.S., Cho I.H., Kim Y.S., Heo S.J., Kwon H.B., Lim Y.J., *Bone-implant interface with simulated insertion stress around an immediately loaded dental implant in the anterior maxilla: a three-dimensional finite element analysis*. Int J Oral Maxillofac Implants, 2012. 27(2): pp. 295–302.



- [19] Liang R., Guo W., Qiao X., Wen H., Yu M., Tang W., Liu L., Wei Y., Tian W., *Biomechanical analysis and comparison of 12 dental implant systems using 3D finite element study*. Comput Meth Biomech Biomed Eng, 2015. 18(12): pp. 1340–8.
- [20] Desai S.R., Desai M.S., Katti G., Karthikeyan I., *Evaluation of design parameters of eight dental implant designs: a two-dimensional finite element analysis*. Niger J Clin Pract, 2012. 15(2): pp. 176–81.
- [21] Huang Y.M., Chou I.C., Jiang C.P., Wu Y.S., Lee S.Y., *Finite element analysis of dental implant neck effects on primary stability and osseointegration in a type IV bone mandible*. Biomed Mater Eng, 2014. 24(1): pp. 1407–15.
- [22] Lombard M., *SolidWorks surfacing & complex shape modeling bible*, 2013: Wiley.
- [23] Verri F.R., Batista V.E., Santiago J.F., Jr., Almeida D.A., Pellizzer E.P., *Effect of crown-to-implant ratio on peri-implant stress: a finite element analysis*. Mater Sci Eng C Mater Biol Appl, 2014. 45: pp. 234–40.
- [24] Winter W., Klein D., Karl M., *Effect of model parameters on finite element analysis of micromotions in implant dentistry*. J Oral Implantol, 2013. 39(1): pp. 23–9.
- [25] Ramos Verri F., Santiago Junior J.F., de Faria Almeida D.A., de Oliveira G.B., de Souza Batista V.E., Marques Honorio H., Yoshito Noritomi P., Piza Pellizzer E., *Biomechanical influence of crown-to-implant ratio on stress distribution over internal hexagon short implant: 3-D finite element analysis with statistical test*. J Biomech, 2015. 48(1): pp. 138–45.
- [26] Roateși I., Hutu E., Ghergic D.L., *Dental implant modeling by FEM*. Roman J Stomatol, 2014. 60(4): pp. 211–17.
- [27] Shigemitsu R., Yoda N., Ogawa T., Kawata T., Gunji Y., Yamakawa Y., Ikeda K., Sasaki K., *Biological-data-based finite-element stress analysis of mandibular bone with implant-supported overdenture*. Comput Biol Med, 2014. 54: pp. 44–52.
- [28] Chang C.L., Chen C.S., Huang C.H., Hsu M.L., *FEA of the dental implant using a topology optimization method*. Med Eng Phys, 2012, 34(7): pp. 999–1008.
- [29] Roateși I., Hutu E., Ghergic D.L., *Stress and strain calculation in implant and surrounding bone by FEM*. Roman J Stomatol, 2014, 60(4): pp. 227–34.
- [30] Piccioni M.A.R.V., Campos E.A., Saad J.R.C., de Andrade M.F., Galvão M.R., Rached A.A., *Application of the finite element method in dentistry*. RSBO Revista Sul-Brasileira de Odontol, 2013, 10(4): pp. 369–77.
- [31] Quaresma S.E., Cury P.R., Sendyk W.R., Sendyk C., *A finite element analysis of two different dental implants: stress distribution in the prosthesis, blunt, implant, and supporting bone*. J Oral Implantol, 2008, 34(1): pp. 1–6.
- [32] Dee K.C, Puleo D.A., Bizios R., *An introduction to tissue-biomaterial interactions*, 2002: Wiley-Liss, John Wiley & Sons, Inc., Publication.

- [33] Geng J.P., Yan W., Xu W., *Application of finite element analysis in implant dentistry*, 2008: Springer.
- [34] Cheng H.Y., Chu K.T., Shen F.C., Pan Y.N., Chou H.H., Ou K.L., *Stress effect on bone remodeling and osseointegration on dental implant with novel nano/microporous surface functionalization*. J Biomed Mater Res A, 2013, 101(4): pp. 1158–64.
- [35] Borcic J., Braut A., *Finite element analysis in dental medicine*. 2012: Intech Publication.
- [36] Raabe D., Harrison A., Alemzadeh K., Ireland A., Sandy J. *Capturing motions and forces of the human masticatory system to replicate mastication and to perform dental wear experiments*. in *24th International Symposium on Computer-Based Medical Systems (CBMS)*, 2011: IEEE.
- [37] O'Mahony A., Bowles Q., Woolsey G., Robinson S.J., Spencer P., *Stress distribution in the single-unit osseointegrated dental implant: finite element analyses of axial and off-axial loading*. Implant Dent, 2000, 9(3): pp. 207–18.
- [38] Geng J.P., Tan K.B., Liu G.R., *Application of finite element analysis in implant dentistry: a review of the literature*. J Prosthet Dent, 2001, 85(6): pp. 585–98.
- [39] Morneburg T.R., Proschel P.A., *Measurement of masticatory forces and implant loads: a methodologic clinical study*. Int J Prosthodont, 2002, 15(1): pp. 20–7.
- [40] Huthmann S., Staszky C., Jacob H.G., Rohn K., Gasse H., *Biomechanical evaluation of the equine masticatory action: calculation of the masticatory forces occurring on the cheek tooth battery*. J Biomech, 2009, 42(1): pp. 67–70.
- [41] El-Mekawy N., Fouad M.M., El-Hawary Y.M., Al-Shahat M.A., El-Gendy R., *Scanning electron microscopy observations of osseointegration failures of dental implants that support mandibular overdentures*. Implant Dent, 2013, 22(6): pp. 645–9.
- [42] Roateși I., *Three dimensional finite element method modeling of dental implants biomaterials*. Roman J Mater, 2015, 45(3): pp. 282–89.
- [43] Khorshid H.E., Hamed H.A.F., Aziz E.A., *The effect of two different immediate loading protocols in implant-supported screw-retained prostheses*. Implant Dentistry, 2011. 20(2): pp. 157–66.
- [44] Kumar Y.S., Pant B., Darunkumar Singh K., *Thickness effects on maximum von-Mises stress of a cement mantle in total hip replacement — a finite element study*. J Appl Biomater Biomech, 2009, 7(2): pp. 111–5.
- [45] Keyak J.H., Rossi S.A., *Prediction of FEMoral fracture load using finite element models: an examination of stress-and strain-based failure theories*. J Biomech, 2000, 33(2): pp. 209–14.
- [46] Spyarakos C.C., *Finite element modeling*. 1994: WVU Press.

- [47] Bălăcel E., Lăcătușu Ș., Topoliceanu C., Bălăcel I., Iovan G., Ghiorghe A., *Mathematics model analysis of biomechanical behaviour of three dental materials to loading related to bruxism*. Roman J Oral Rehab, 2011, 3(3): p. 26.
- [48] Lin C.L., Kuo Y.C., Lin T.S., *Effects of dental implant length and bone quality on biomechanical responses in bone around implants: a 3-d non-linear finite FEA*. Biomed Eng Appl Basis Commun, 2005, 17: pp. 44–9.
- [49] Van Staden R.C., Guan H., Loo Y.C., Johnson N.W., Meredith N., *Comparative analysis of two implant-crown connection systems—a finite element study*. Appl Osseointegration Res, 2008, 6: pp. 48–53.
- [50] Misch C.E., *Contemporary implant dentistry*. 2007: Elsevier Health Sciences.
- [51] Cicciu M., Bramanti E., Cecchetti F., Scappaticci L., Guglielmino E., Risitano G., *FEM and von Mises analyses of different dental implant shapes for masticatory loading distribution*. Oral Implantol (Rome), 2014. 7(1): pp. 1–10.