A COMPARATIVE NUMERICAL STUDY OF DENTAL IMPLANTS

MORARU Emanuel-Ioan
Facultatea de Inginerie Industrială și Robotică, Specializarea: Siguranța și Integritatea Structurilor, Anul de studii: Masterat 2, e-mail: emanuelioan.moraru@gmail.com

Conducător ştiinţific: Conf. Dr. Ing. **Emil NUȚU**

ABSTRACT: In the present study, the goal was to create a finite element model of two different dental implants by using 3D Scanning to accurately reproduce the geometrical features of the implants. These models were created to obtain preliminary results on the stress distributions in the two implants to identify the maximum stress zones by applying an axial force with a value taken from the literature. The main drive to do this initial study is to prepare a more complex model of the dental implant system which will take into consideration the mechanical properties of the edentulous and dentate bone with its different zones (cortical and trabecular).

KEYWORDS: dental implants, finite element, 3D Scanning, NASTRAN, FEMAP

1. Introduction

The present study has the main goal to model two dental implants, starting from 3D scanning, by using a finite element method (FEM) preprocessor and to determine the stress distribution into implants by means of a static structural analysis with an axial load on the implant, emulating the occlusal load.

The stages applied for achieving the CAD (computer Aided Design) models of the implants for subsequent FEM evaluation consist of 1- acquiring the dental implants, 2- reconstruct their virtual geometry by 3D scanning means, 3- prepare the geometry using Computer Aided Manufacturing (CAM) software (Autodesk ReCap Pro and Autodesk Fusion 360) and 4-then import the geometry file in a FEM preprocessor to model it. The software used for the FEM pre- and post- processing was FEMAP v12.0.1 and the solver used for generating the results was NASTRAN 2019 FP1.

To create a trustable finite element model, adapted to clinical reality, a literature study was done regarding the medical terms, the phenomenological aspects in the implant region and other FEM studies. In this respect, the main papers that were reviewed were published in medical and biomechanical journals.

The last goal was to prepare a more complex model plan to continue this study by modelling the implant system using bone regions and computer tomographic images (CT) to reduce the errors given by the approximations used in this study. This last goal was achieved by studying different publications regarding the mechanical properties of edentulous and dentate human bone structures with or without an implant system present. This plan refers also to using non-conventional modelling of the affected areas by using element types specific to NASTRAN.

2. The dental implant

A dental implant is a treatment solution for replacing missing teeth (this pathology being named edentulism). The dental implant is a prosthetic structure made of alloplastic materials (materials which are biocompatible but are not naturally present in the human body). It will be implanted in the oral tissue beneath the mucosa and periosteum, through the bone for retention and support of the crown or prosthesis.

The common implant prosthesis has three main components: the implant (the screw which will be implanted in the oral tissue), the abutment and the dental crown, as it is presented in figure 1.

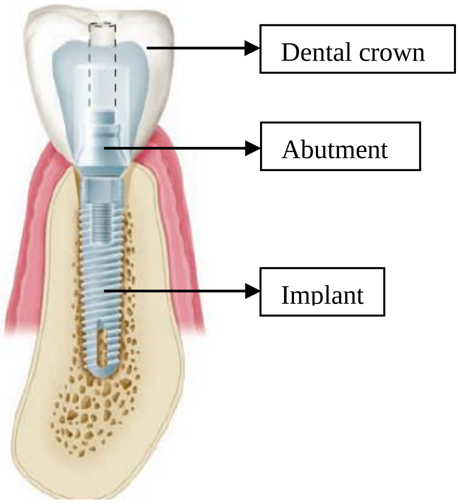

Fig. 1 **Implant assembly components**

The most common biocompatible materials used in nowadays dentistry are Titanium alloys, hydroxyapatite, zirconium, and sapphire bio ceramics. Zirconium is commonly used for the dental crowns, for the abutments, titanium alloys, while the implants are mostly made of titanium alloy.

For each of the three parts constituting the implant assembly (figure 1), there are lots of variations in shape. In the following, a classification taken from paper [1] is presented. The four main implant types are presented in Figure 2. Implant type A is a screw implant made of titanium alloy with no surface treatment, Implant type B is a titanium alloy screw implant plasma coated, type C is a cylinder implant coated with hydroxyapatite and the last type of implant (D) is a cylindrical titanium alloy implant plasma coated.

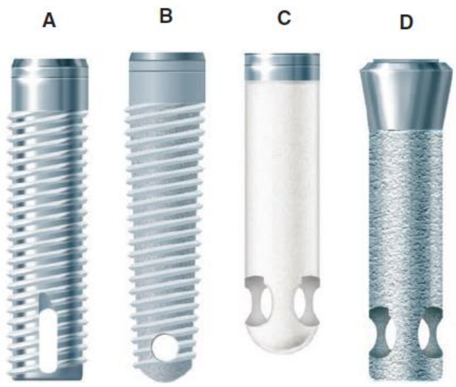

Fig. 1**. Implant classification**

There are also several types of abutments. These different abutments, as the different types of implants are used for different pathologies for the edentulous patient. These pathologies can be given by topological reasons (the part of the maxilla/mandible bone in which the implant will be inserted) or by different diseases regarding the bones or gums.

According to [1] the different types of abutments are presented in Figure 3, as follows: A is a standard abutment used for a simple screwed dental crown (the connection of the crown to the implant is made by a screw, which will ensure the necessary mechanical and functional support), B is a fixed abutment which will be used for attaching a cemented crown (a crown mounted on an implant by means of an adhesive – dental cement), C is a angled abutment, which can be used for both of the dental crown configurations spoken about above, D is a conical abutment used for the screwed dental crowns and E is a nonsegmental or direct abutment used for screw retained dental crowns.

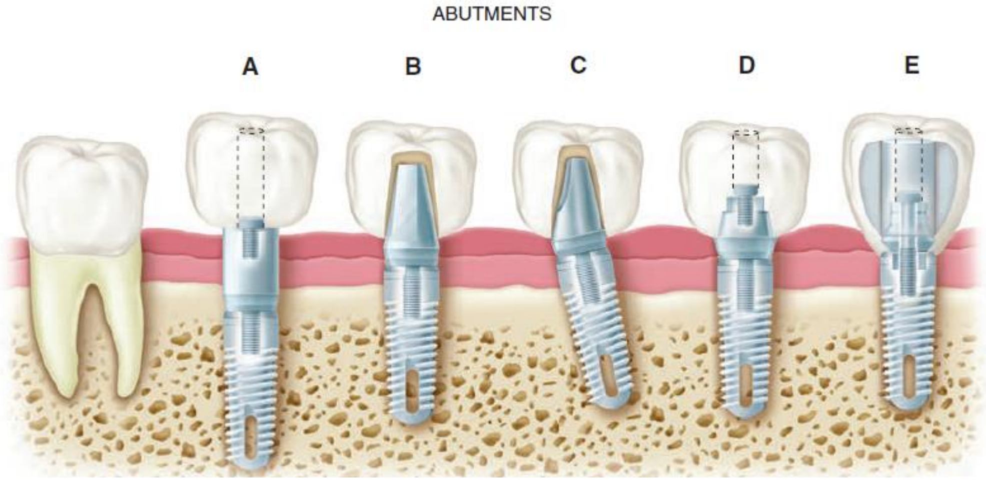

Fig. 3. **Abutment types**

Figure 4 presents a visual explanation of the two types of dental crowns. The screw retained dental crown is mounted onto the abutment by means of a screw, while the cemented dental crown is mounted onto the abutment by means of an adhesive substance and the abutments are screwed into the implant.

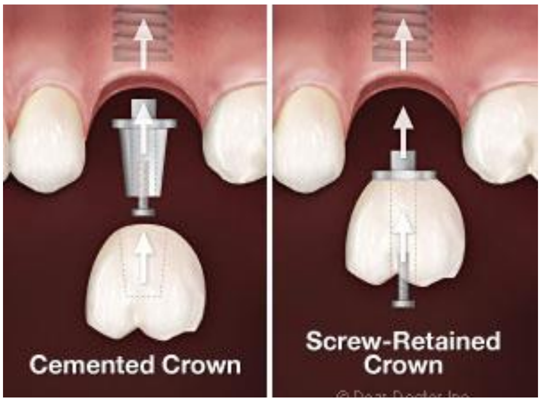

Fig. 4. **Dental crown types**

3. FEM studies for the dental implants

The start of the study was to find a paper regarding the subject to be able to see the methodology of modelling the implant and to compare the information regarding the physiology of the dental occlusion (the way a human closes the mouth and the forces which appear in the process of mastication). The most relevant study found was paper [2]. In this study, the main goal was to determine the influence of deformation and stress between the bone and implant assembly from various bite forces using numerical simulation analysis. The first step the authors made was to take a part of an edentulous mandible (corresponding to the first molar, given the fact that this region is mostly loaded during mastication) from a CBCT (cone-beam computed tomography) scan and model it as a homogenous part with the density and mechanical properties of a cancellous bone. In Table 1 the values of the mechanical properties used in paper [2] are given. The next step was to model the assembly between the whole implant system and the bone, and this was done using a CAD program.

The loading of the system was made at more angles (0, 90, 30, 45 and 90 degrees) and the forces varied in the interval 100-500 N. They successfully managed to obtain stress and displacement distributions under considerable simplifications of the model (the bone was modeled as homogenous, and linear bonded contact between the parts of the assembly was applied).

Tabel 1. Material properties of the implant system [2]

Component/material	Crown/Zirconia	Abutment/Titanium	Implant/Titanium	Spongy bone
Young modulus [GPa]	200	96	96	1.37
Pisson's ratio	0.3			

In the study presented here, the first step was to perform a static linear analysis of the implant without the bone using the same values for the loading as in study [2] to identify the critical area of the implant, from the strength perspective.

3. The numerical models

The implants used in this study are RESISTA implants with different configurations. Given the fact that the CAD files of the implants were not provided, to ensure that the geometrical properties of the implants were kept, the implants were 3D scanned by means of a intrabuccal 3D scanner used in dental practice. These scanners are not very precise regarding the fillet of the implants, but for a preliminary numerical study, the results obtained were good enough.

After the scanning, AUTODESK ReCap Pro (Student license) was used to clean the mesh obtained and then the obtained mesh was imported in FEMAP for the FEM preprocessing.

In the Figure 5, the implant configurations are presented (only the External Hex Connection and the Uni Q MUA configurations were analyzed), while in Figure 6 are presented the scanning device and the resulting virtual geometries of the scanned implants.

Fig. 5 **Implant configurations analyzed within this paper**

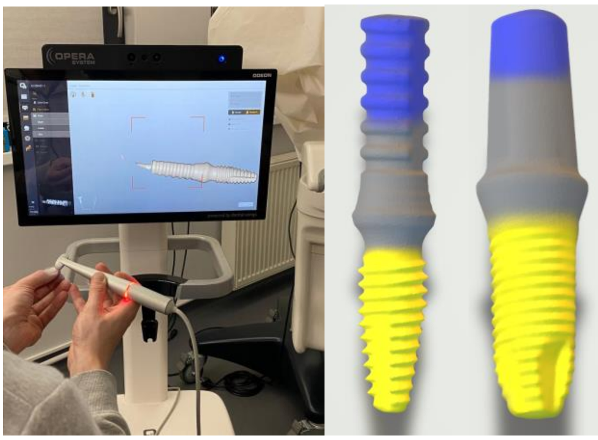

Fig. 6 **3D Scanning using an intrabuccal scanner and the resulting implant CAD models**

For the FEM modelling the material properties used for the implant are taken from [2].

The boundary conditions applied in this study consist of blocking the translations on the screw part of the implant and applying a 1000N force on the implant head (maximal statistic force of mastication in the first molar zone). In Figure 7 there are highlighted with yellow the regions were translations were blocked and with blue the application regions of the force, for the External Hex Implant (left) and the UNI Q MUA Implant (right). The yellow zone is the one considered to be inserted into the bone while the blue zone represents the zone on which the abutment-crown system presses.

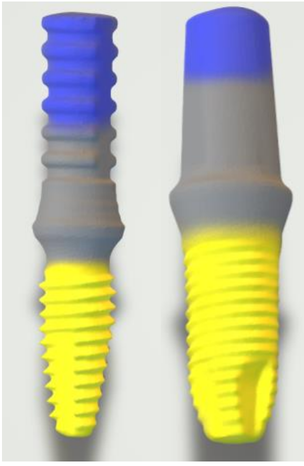

Fig. 7. **The boundary conditions applied on the two analyzed implants: restraints in al translations (yellow regions) and mastication forces (blue regions)**

4. Results

Using NASTRAN, the results were obtained and visualized with FEMAP. As expected, the stresses resulted are high because the analysis performed is a static linear analysis and there is no bone structure to dampen the load. The constraints are rigid. In Figures 8 and 9, the von Mises stresses (left) and the total displacements (right) are plotted for the two implants respectively.

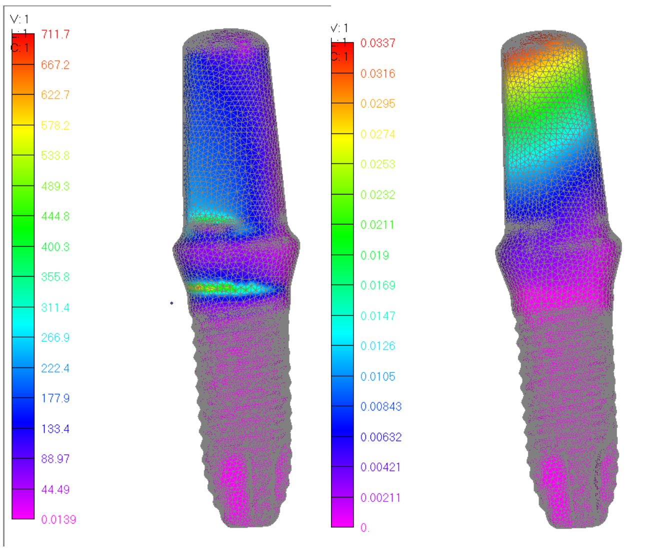

Fig. 8. **von Mises stresses (left) and total displacements (right) in the Extrenal Hex Implant (Units: MPa and mm)**

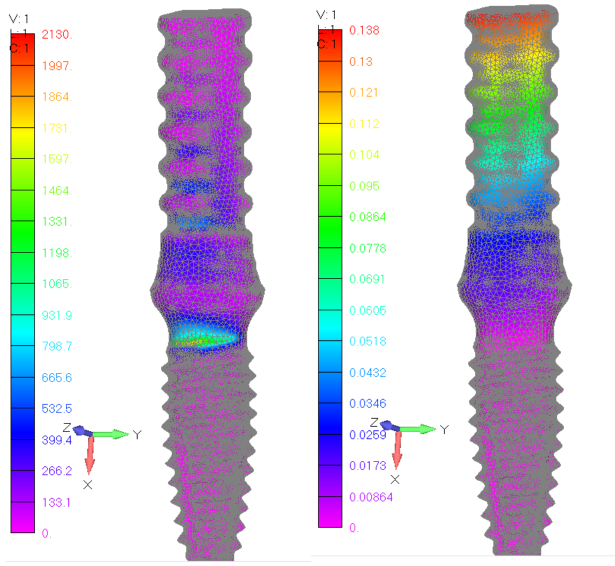

Fig. 9. **von Mises stresses (left) and total displacements (right) in the UNI Q MUA Implant (Units: MPa and mm)**

5. Conclusions

It can be concluded that the maximum loaded zone of the two implants is the flange zone at the interface between the implant and the bone.

The 3D scanning helped preserving the geometry and is a viable tool to be used when dealing with a complex geometry in analysis.

The further research focuses on modelling the bone and the hole into the implant head to obtain more realistic values of stresses and displacements for the implants and make physical tests to be able to check if the assumptions made in the FEM model are close to the reality.

8. Bibliography

[1]. Rosenstiel, S., Contemporary Fixed Prosthodontics 5th Edition, Mosby, 2015, ISBN 0323080111

[2]. Cheng H.C., Peng B.Y., Chen M., Hunag F., Lin Y., Shen Y., Influence of Deformation and Stress between Bone and Implant from Various Bite Forces by Numerical Simulation Analysis, Biomed Res Int. 2017; 2017: 2827953.

[3]. Fazi G, et al: Three-dimensional finite element analysis of different implant configurations for a mandibular fixed prosthesis. Int J Oral Maxillofac Implants 26:752, 2011.

[4]. Weinberg LA: The biomechanics of force distribution in implantsupported prostheses. Int J Oral Maxillofac Implants 8:19, 1993.

[5]. Kent J, et al: Biointegrated hydroxlapatite-coated dental implants: 5-year clinical observations. J Am Dent Assoc 121:138, 1990.

[6]. Smith D, Zarb GA: Criteria for success for osseointegrated endosseous implants. J Prosthet Dent 62:567, 1989.

[7]. *** MSC Nastran 2021.2, Quick Reference Guide