MŰEGYETEM 1782

BUDAPEST UNIVERSITY OF TECHNOLOGY AND ECONOMICS
DEPARTMENT OF STRUCTURAL MECHANICS

The mechanical behavior of the bone microstructure around dental implants

Summary and Theses of PhD dissertation

ÉVA LAKATOS

Supervisor:

DR. IMRE BOJTÁR

Budapest, 2011.

CONTENTS

1. INTRODUCTION

Dental implantation is currently the most commonly used and physiologically the most favorable procedure for tooth replacement in dental surgery. Implants can have either advantageous or destructive effect on the surrounding bone, depending on several physiological, material and mechanical factors. In the light of this, implants should be applied, that transfer occlusal forces to bone within physiologic limits and have geometry capable to enhance bone formation. To this, stress and strain distributions around different types of dental implants need to be assessed. The most general method for estimating the biomechanical behavior of the bone is finite element analysis (FEA).

In the last few years efforts have been made to decrease peak stresses and to obtain more even stress distribution by: choosing more favorable thread formation, increasing the length or the diameter of the implant or controlling the method of load transfer from the prosthetic component to the implant. In most reported studies the macroscopic geometry of the implant is modeled in FEA and the assumption is made, that the materials are homogeneous and have linearly elastic material behavior characterized by two material constants of Young's modulus and Poisson's ratio. At the bone-implant interface most FEA models assume optimal osseointegration, meaning that cortical and trabecular bone is supposed to be perfectly bonded to the implant.

Bone mineral density, geometry of bone, microarchitecture of bone, and quality of the bone material are all components that determine bone strength as defined by the bone's ability to withstand loading. For this reason, microstructural information must be included in the analysis to predict individual mechanical properties of bone. Microstructural modelling of trabecular bone has become an extensively investigated field of biomechanical researches nowadays. The most commonly used tool for characterizing the complex architecture and material properties of bony organs is the conversion of micro-computed tomography images into micro-FE models.

Within the framework of my PhD study I deal with the finite element modeling of the bone surrounding dental implants in special consideration of the microstructure of the cancellous bone. The research was created in close cooperation with the Faculty of Dentistry, Semmelweis University, Budapest. The purpose of the study was to create a new, numerical bone modeling method that promotes the investigations of the bone tissue surrounding dental implants with mechanical properties comparable to that of the real bone tissue and that – combined with finite-element modeling of screw-type implants – is applicable for simulating the mechanical behavior of the cancellous bone faster and simpler than others in the literature. I also aimed to obtain a deeper knowledge about the trabecular substance of the jawbone.

In the dissertation I deal with the following topics:

- the material properties and the microstructural modeling of the trabecular bone,
- the modeling of the cortical bone,
- the implant and
- the bone-implant interface with incomplete bonding,
- compilation of the complex model and
- the illustration of the applicability of the model in the design of an implant group, which is currently under design and is intended to be applied in everyday dental surgery.

2. METHODS AND RESULTS

2.1. EXPERIMENTAL DETERMINATION OF THE MANDIBULAR TRABECULAR BONE MATERIAL PROPERTIES

2.1.1. YOUNG-MODULUS MEASUREMENTS OF THE TRABECULAR JAWBONE BY MEANS OF COMPRESSION TEST

The aim of the following experiments was to determine the mechanical properties of the human mandibular trabecular bone, to be used in further finite element models. Therefore compression tests were conducted in the laboratory of the Research Center for Biomechanics of Budapest University of Technology and Economics. The bone specimens were provided by the Department of Forensic Medicine of Semmelweis University.

The experiments were performed on fresh and macerated cadaveric samples. Each specimen was taken from the molar mandibular region of middle aged male patients from the lower edge of the bone. Since we aimed to examine the trabecular bone, the cortical layer around it was cut, the way it is shown in *Figure 1*.

The samples were submitted to compression tests using a Zwick Z005 displacement controlled testing machine and force-displacement pairs were registered. *Figure 2* shows a typical example of the received force-displacement curves, which corresponds the characteristic diagram of the compressed cellular solids. The initial, closely linearly elastic part comes from the elastic bending of the trabeculae, the long horizontal plate shows the gradual failure of the spongiosa, until the cell walls touch and the curve increases steeply.

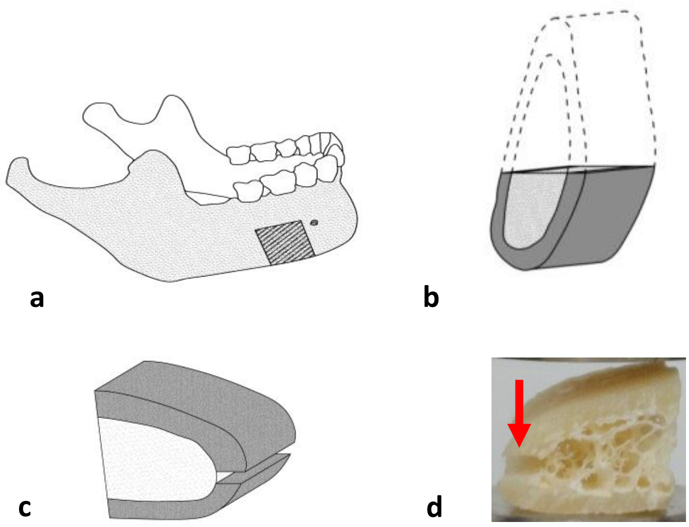


Figure 1. The position of the bone specimens in the mandible (a) and its cross section (b), the illustration of the cut cortical bone (c-d)

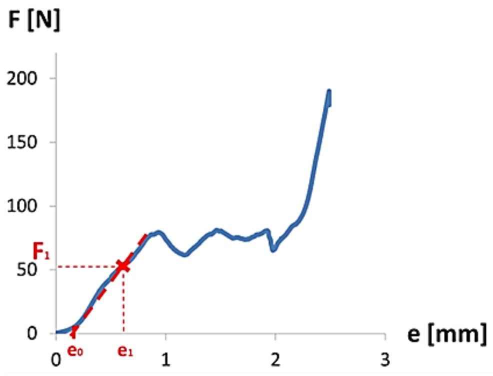


Figure 2. Force-displacement curve of the trabecular bone in the mandible under compression

Since the geometry of the specimens was complex and varying, the Young's modulus of the trabecular bone could not be determined directly. For the computer simulations of the experiments a finite element model (*Figure 3*) was created.

The geometrical parameters of the model were set according to the specific specimen. Both the cortical and the trabecular bone were assumed to be linearly elastic continuums. The material properties of the cortical layer were set according to data from literature. The Young's modulus of the trabecular bone was determined by simulating the compression test: loading the upper side of the model with vertical force and constraining the lower side against horizontal and vertical displacements. An arbitrary force (F_1) value from the initial elastic part of the F-e diagram was applied and the Young's modulus of the spongiosa was chosen to result in the same displacement (e_1-e_0) as in the compression test (where e_0 is a displacement from the initial balancing, resulting from the inaccuracy of the specimen geometry) using an iteration algorithm.

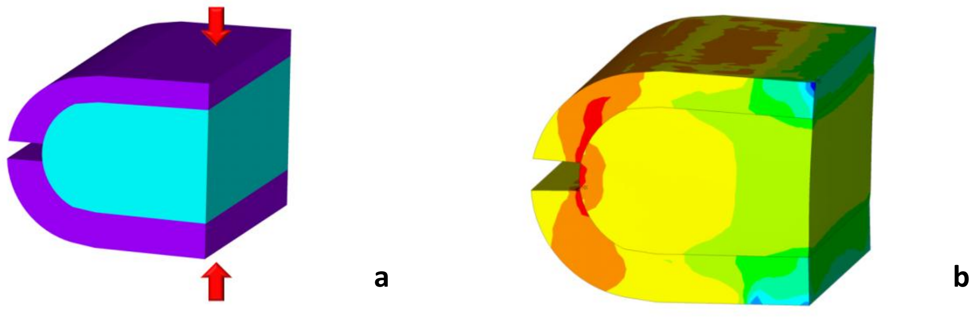


Figure 3. Specimen specific finite element model for Young's modulus calculations (a) and the vertical normal stress distribution from the vertical compressive load (b)

2.1.2. ANISOTROPY MEASUREMENTS IN THE TRABECULAR JAWBONE USING MICRO-CT IMAGES, BY MEANS OF INSERTED ELLIPSOIDS

Because of its orthotropic mechanical properties the anisotropy of the trabecular bone can be characterized by three main directions and the characteristic values of their dominances. The anisotropy of the orthotropic architecture can be described by fabric tensors and their 3×3 matrices, where the eigenvectors of the tensors mean the main directions and the corresponding eigenvalues give the dominances. The importance of fabric tensors concerning the anisotropy of the trabecular bone resides in the fact, that fabric and the mechanical properties are related.

A new method has been developed for the micro-CT based anisotropy measurements of the trabecular bone. The basic principle is to find the largest ellipsoids around certain points in the material, which contain bone and their surfaces touch the medullary cavity (*Figure 4*). Each ellipsoid is obtained by the gradual enlargement of an initial sphere.

The fabric tensors of the certain points can be compiled using the main axes of the inserted ellipsoids and their average gives the fabric tensor of the structure.

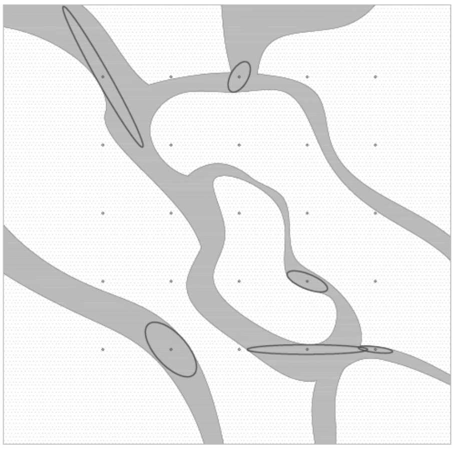


Figure 4. Illustration of the new method developed for the measurement of the anisotropy of porous structures using inserted ellipsoids

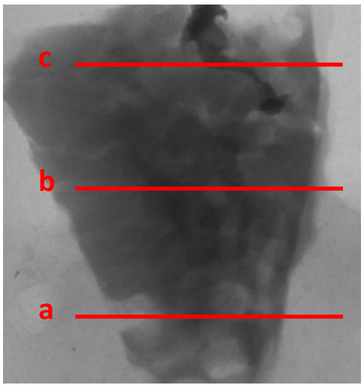
ANISOTROPY OF THE BONE AROUND THE HUMAN TOOTH ROOT

The anisotropy and porosity of the trabecular bone were examined using bone samples, which were obtained from living human in medically justified dental surgical operations. The specimens were submitted to micro-CT scanning and the orientation of the bone was determined in the following steps:

Sampling:

- Excision of the bone samples during medically justified dental surgical interventions.
- Marking the samples with gutta-percha (a material used to obturate, or fill the empty space inside the root of a tooth after it has undergone endodontic therapy), which gives X-Ray shadow on the CT images.
- Setting down the exact place of the marks.

Micro-CT scanning:



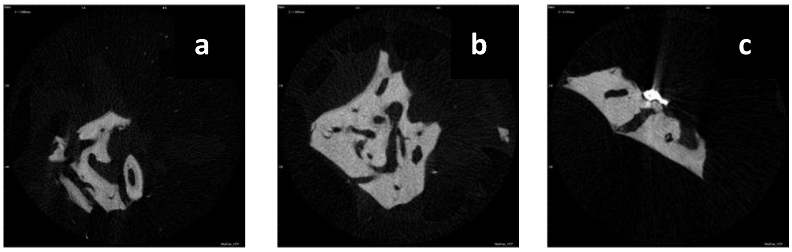


Figure 5. Slices of the micro-CT images (the bone material is denoted by light grey color and the bright white indicates the gutta-percha mark) (c)

Preparing the data base to be used for the measurements:

- Creating the three dimensional matrix, that contains the linear attenuation coefficients of the material point by point by linking together the CT image slices.
- Binarisation of the data base.
- Identification of the gutta-percha marks in the data base.
- Cutting out a purely trabecular domain from the matrix representing an irregular shaped bone sample.

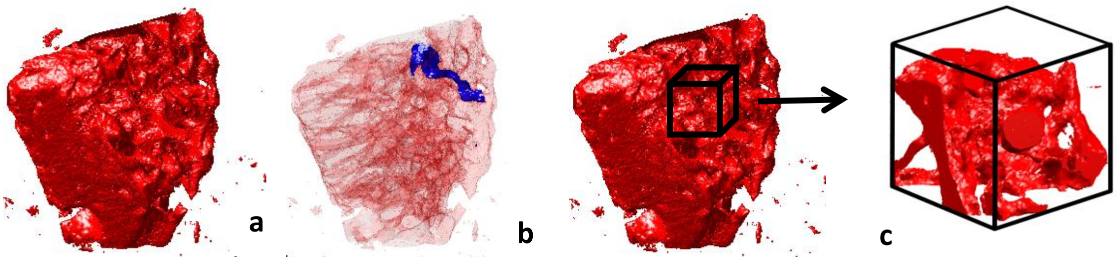


Figure 6. Preparing the data base: binarisation (a), identification of the gutta-percha marks (b), cutting out a purely trabecular domain (c)

Porosity calculations:

One from the 10 samples proved to be too dense (very low porosity) with no clearly observable trabecular microstructure, so it had to be excluded from the further examinations. The remaining 9 showed 18%–74% porosity values.

Anisotropy calculations using inserted ellipsoids:

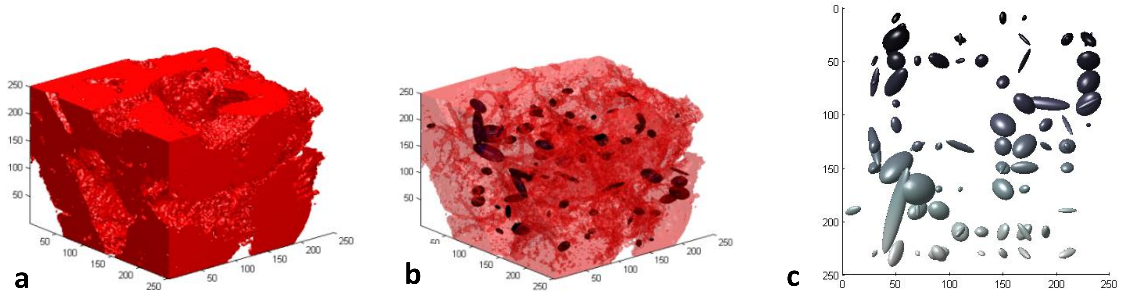


Figure 7. Inserted ellipsoids: a bone domain (a) bone with ellipsoids (b), ellipsoids without bone (c)

Comparing the orientations with the physiologically predictable load directions:

Comparable results could be achieved by transforming the fabric tensors and the dominant directions in a uniform, anatomic coordinate system. Figure 8 shows the dominant directions (the first eigenvectors of the fabric tensors) of the certain samples represented around an idealized tooth root.

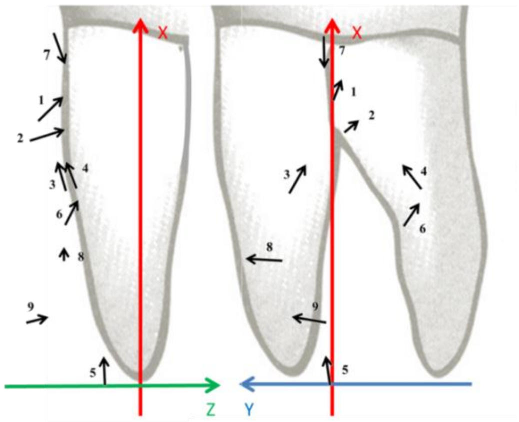


Figure 8. The first eigenvectors of the fabric tensors of the certain samples represented around an idealized tooth root

From the measurements it can be concluded, that the trabecular bone around living tooth possesses anisotropic geometrical properties and it is dominantly directed about the vertical (*X*) direction.

2.2. SIMULATIONS OF THE LOAD DEPENDENT ANISOTROPY OF THE TRABECULAR BONE USING FINITE ELEMENT BEAM MODELS

The following part of my dissertation deals with the microstructure of the bone. The aim of the research was to create a finite element model of the trabecular bone, which uses the results of less patient specific medical examinations with less radiation exposure than CT scanning as input data (e.g. bone density).

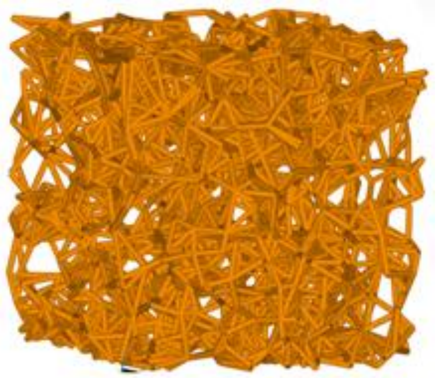


Figure 9. Finite element beam model of the trabecular bone

The initial objective was to create a program that generates a stochastic beam structure, which has the format identical with the input data of program system ANSYS and has parameters, which are revisable according to the bone that is simulated (in the aspect of density, porosity, elastic properties etc.).

The finite element beam model of the cancellous bone was created by interlinking a stochastically generated set of nodes in a certain domain according to a previously defined linking-rule. In the received model each trabecula is represented by one beam element (*Figure 9*).

Bone tissue is continuously remodeled through the processes of bone formation and bone resorption; this renewal is called functional adaptation. Bone remodeling is generally viewed as a material response to the continuously changing load conditions we experience during our lifetime: increasing loads cause bone growth, while decreasing loads cause bone resorption. The coupled apposition and resorption are regulated through feedback mechanisms at the cellular level as a function of the local mechanical stimulus leading to a change in the arrangement of the trabeculae (*Figure 10*). The remodeling process leads to a configuration of the trabeculae in the cancellous bone, in which they follow the directions of the principal stress trajectories.

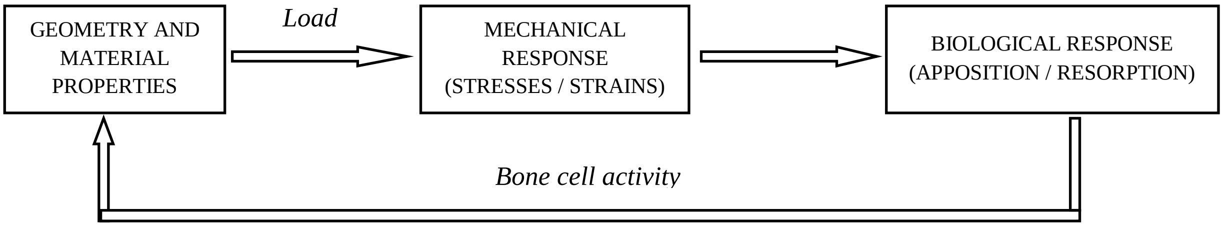


Figure 10. A simplified scheme of the load directed bone remodeling

For the purpose of simulating the remodeling of the porous, cancellous structure of the trabecular bone tissue, a frame model was used based on the aforementioned stochastic mesh, but containing more beam elements. In the later examinations each node is connected to twice as many (i.e. 14) neighboring nodes as in the original (anatomical) configuration. Only the number according to the original model, however, has a load-bearing role. These elements are regarded as ‘active’ and possess the Young’s modulus value of the original model. The stiffness of the other elements in the structure is decreased in order to minimize their load-bearing function to a negligible level compared to the working elements. These are called ‘passive’ elements, and have a much lower (three orders of magnitude) stiffness. The configuration in accordance with a certain load is determined by the beams being ‘active’ or ‘passive’. To maintain the original geometrical conditions, the basic requirement is to keep the number of the active node connections at its original value for each node during the loading process.

Based on the concept outlined above, two different stress-dependent bone remodeling algorithms has been developed, one that gives an ideal configuration to a given static load and one which is capable of following the loading process and altering the bone structure accordingly. The maximal normal stress which arose in each trabecula was used as the mechanical stimulus for bone adaptation. The basis of the method is to assign two weighted fabric tensors – one for tension and one for compression – to each node according to the stresses transferred from the connecting beams to the given node.

The elimination of the isotropy and the development of the new, trajectory directed orientation belonging to a certain load were examined by means of fabric tensors and orientation distribution functions of the beams, under compression and shear load (*Figure 11*).

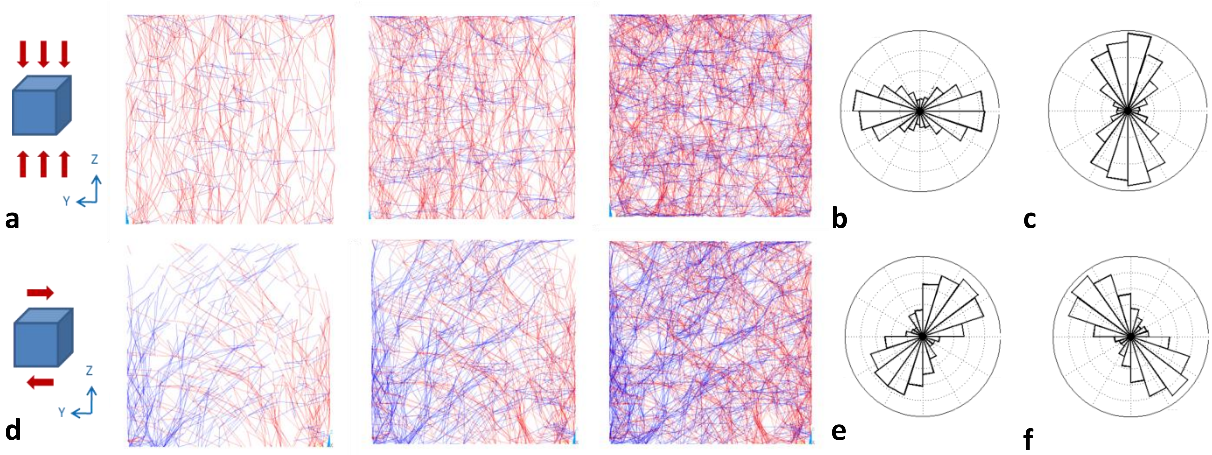


Figure 11. Illustration of the remodelling algorithms: The development of the trajectory directed orientation in the frame structure from static vertical, unidirectional compressive (a, b, c) and shear (d, e, f) load and the angular distribution of the tension beams (b és e) and the compression beams (c és f)

2.3. MECHANICAL BEHAVIOR OF THE BONE AROUND DENTAL IMPLANTS

The success of a dental implant depends on various conditions. By means of finite element analysis the stress and strain distribution around a healed implant under everyday occlusal loads, so the question of load transfer can be examined effectively. The distribution of stresses in the bone around an implant can be determined by the complex model containing the implant, the cortical and the trabecular bone together.

2.3.1. FINITE ELEMENT MODEL OF THE SCREW TYPE IMPLANT

To make the everyday dental surgical design easier, I developed a method for the modelling of screw type dental implants by means of mathematical functions and several modifiable or revisable parameters, which makes the procedure of finite element modeling faster and easier *(Figure 12)*. The parameters, which can be altered according to the sizes and shapes of the implants chosen by the dental surgeon, are the following: the length of the implant, the inner and outer diameters of the implant, the thread formation and thread pitch, all of which can change along the length according to a function or differ in certain segments and the shape of the implant head and apex.

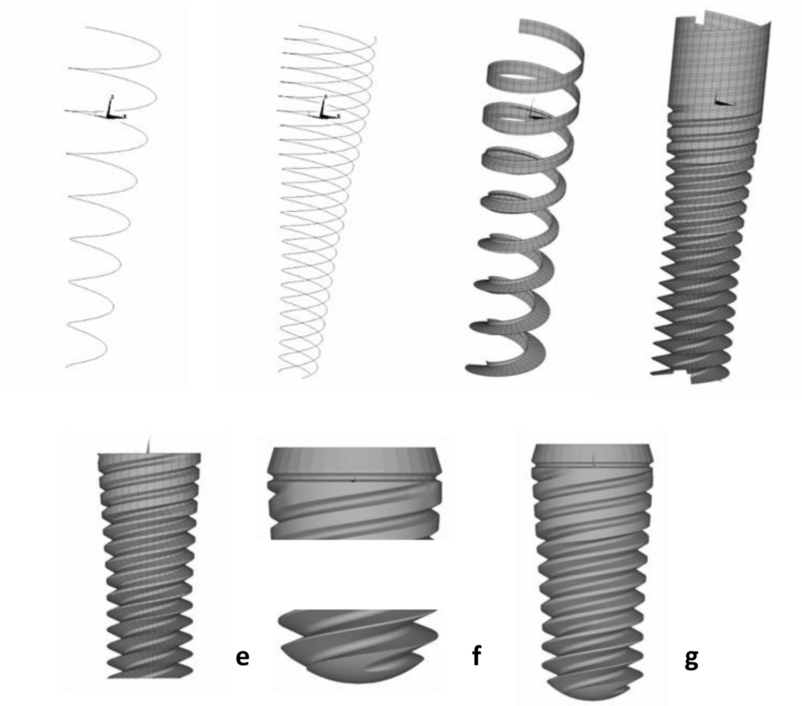


Figure 12. The building of the finite element model of a screw type dental implant: the spiral director curve of one thread (a), the director curves of all three threads (b), the thread form (c), the surface of the threads (d), removal of unnecessary parts (e), implant head and apex (f), geometry of the implant (g)

Material properties of dental implants can be approached quite accurately – since being metal alloys – as homogeneous and linearly elastic. In these days the application of metallic materials in dental implantology is limited to commercially pure titanium and its *Ti-6Al-4V* alloy. The finite element mesh of the screw was built from 10 node quadratic tetrahedral elements for the most accurate geometry possible.

2.3.2. FINITE ELEMENT MODEL OF THE BONE

Bone tissue has two different settlements in the human skeleton as well as in the jawbones. The solid, compact substance forms the external, cortical layer and the internal trabecular substance is porous and cancellous. The upper egde of the cortical layer of the edentolous mandible and maxilla often grows thiner. Since the thickness of the cortical bone is an important factor of the successfulness, in the model it was enabled to be changed.

Depending on the aims of the certain examimation the trabecular bone was modelled as a finite element frame, which might as well have remodelled or as a continuum.

2.3.3. COMPILING THE COMPLEX MODEL, BONE-IMPLANT INTERFACE

The complex parametric finite element model of the bone around dental implants divided into several submodels: model of the implant, the cortical and cancellous bone as well as the bone-implant interface and the imperfect connection. The whole model was compiled by intersecting the volume and line elements of the aforementioned submodels (*Figure 13*).

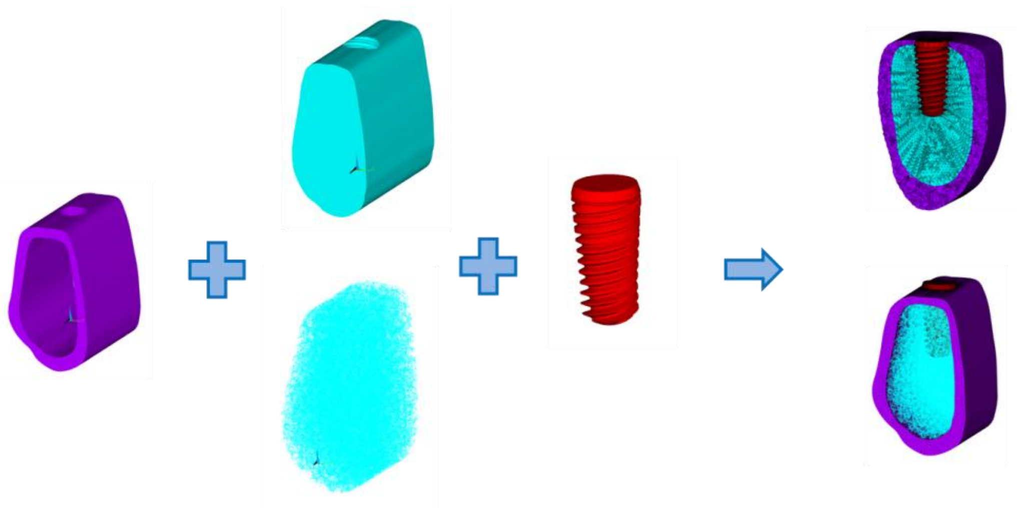


Figure 13. Compiling the complex model

While compiling the complex finite element model using trabecular bone model with beam elements the problem of connecting the different element types had to be solved. The beams cut by the surface of the implant or the cortical bone had to be connected to the nodes of the surface mesh, so an algorithm had to be developed, which changes the end points of the cut beams from the original connection points to the closest nodes of the surfaces (*Figure 14*).

Further problems came up when the beams were cut. Too short elements appeared, and beams that were not connected to any other beams. These elements had to be found and erased.

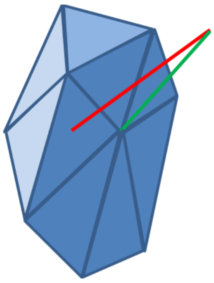


Figure 14. The connection of the beams to the finite element mesh of the surface: the original beam (red) changed to a properly connecting one (green)

Since perfect osseointegration cannot be assumed, the incompletion of the bonding between the bone and the implant was taken into account as a revisable parameter.

2.3.4. APPLICATION OF THE COMPLEX MODEL

The method developed for the modeling of screw type dental implants makes the everyday dental surgical design easier. The screw generating algorithm is effectively utilized in practice in the design of a new group of dental implants (*TRI-Vent Dental Implants, TRI Dental Implants Int. Ag. - Switzerland*). In the forthcoming example the developer requested an answer to the following questions:

- What kind of effect do the micro-grooves and micro-threads at the head region of the implant have on the stress distribution around the implant compared to an implant with smooth surface at the head region?
- The smallest member of the *TRI-Vent Dental Implant* series could not be designed with the physiologically advantageous platform switched head for reasons of strength. Is the applied cylindrical head mechanically disadvantageous?
- What is the ideal depth of screwing depending on the cortical bone thickness?

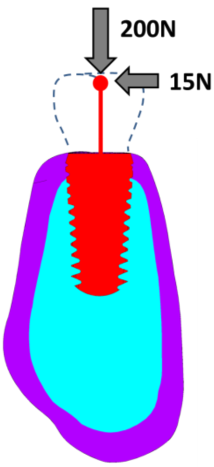


Figure 15. The applied loads

I searched for the answers to the questions by generating the finite element model to each implant geometry recommended by the developer. The stress distribution in the bone was examined under occlusal forces (*Figure 15*). The magnitude and the distribution of the stresses were examined in the trabecular and cortical bone tissue under the various circumstances.

From the simulation results it can be concluded, that the magnitude of the maximum stresses in the bone was the lowest, the stresses were the most evenly distributed and the heavily loaded area was the least extended, when a micro-grooved implant was fastened, (in most cases) leveling the cortical bone surface. The cylindrical head form of the smallest sized implant proved not to be disadvantageous in the aspect of stress distribution in the bone.

3. NEW SCIENTIFIC RESULTS

I summarized my results in the following four theses:

Tesis 1. *I developed a new, simply executable computer simulation aided method for the Young-modulus measurement of the mandibular trabecular bone. I conducted laboratorial tests on cadaveric samples, by means of which a domain of the Young-modulus value of the human mandibular trabecular bone can be given.* [10]

Tesis 2. *I evolved a new numerical method for the measurement of the structural anisotropy of the trabecular bone based on inserted ellipsoids, using micro-CT images. By means of the developed method, using bone samples from living humans I proved that the trabecular bone around the tooth root possess anisotropy. I determined its principal directions and their dominances.* [9]

Tesis 3. *I developed a new method for the finite element modeling of the microstructure of the trabecular bone by means of a finite element frame model* [2, 6, 8]. *For the simulation of the load dependent anisotropy in the trabecular bone two frame model based methods have been developed:*

- *one for determining the ideal configuration of the beams according to a certain load* [4],
- *and one that is capable of following an altering loading process with the bone structure transforming according to the varying conditions based on load dependent fabric tensors* [1].

Tesis 4. *I created a new finite element model of screw type dental implants, which possess revisable geometric parameters. The possibility for the fast and easy modification of the geometric properties helps the design and development of dental implants. The complex parametric finite element model of the bone around dental implants have been compiled by assembling the finite element models of the trabecular bone, cortical bone and the implant, with the incomplete osseointegration taken into account. The clinical application of the complex model have been illustrated through my proposals concerning an implant group, which is currently under design and is intended to be applied in everyday dental surgery* [3, 5, 7].

4. SUMMARY AND PROPOSAL FOR FURTHER RESEARCH

In the dissertation I dealt with the finite element modeling of the bone surrounding dental implants in special consideration of the microstructure of the cancellous bone. The complex research program divided into four parts: the examination of the trabecular bone, the cortical bone, the dental implant and the bone-implant interface. To get acquainted with the structural and mechanical properties of the trabecular bone I performed laboratory tests and utilized the experiences in my further research into the modeling of the microstructure of the trabecular bone and its load dependent anisotropy and into the behavior of the implant imbedded in the bone. I examined the geometric properties of the implant and the effect of the depth of screwing by means of a complex finite element model containing the trabecular and cortical bone, the implant and the incomplete bone–implant interface.

The possible directions in which the research introduced in the dissertation can be continued are the following:

- The effect of the process of screwing on the bone around the implant.
- The effect of local defects in the bone.
- The effect of those anatomic properties, which are measurable in living human on the jawbone quality and on the behavior of the finite element models introduced in the field of oral implantology (e.g. the effect of age, sex, drugs, diseases).

PUBLICATIONS ON THE SUBJECT OF THE THESES

International journal papers

1. **Lakatos É., Bojtár I.:** Trabecular bone adaptation in a finite element frame model using load dependent fabric tensors. *Mechanics of Materials,* **44** (2012) pp. 130-138 (IF in 2009 2,206)
2. **Lakatos É., Bojtár I.:** Stochastically Generated Finite Element Beam Model for Dental Research, *Periodica Polytechnica Ser. Civ. Eng.*, **53/1** (2009), pp. 3-8 (IF 0,222)
3. **Lakatos É., Bojtár I.:** Microstructural simulations of the bone surrounding dental implants by means of a stochastically generated frame model, *Biomechanica Hungarica*, **3/1** (2010) pp. 143-150

Hungarian journal papers

4. **Lakatos É., Bojtár I.:** Simulations of the bone remodeling by means of stochastically generated finite element frame models *(in Hungarian)*, *Biomechanica Hungarica*, **3/2** (2010) pp. 31-41

Monograph

5. **Lakatos É., Magyar L., Bojtár I.:** Material properties of the trabecular bone in the human jaw-bone *(in Hungarian)*, *Építőmérnöki Kar a Kutatóegyetemért*, monograph, published by the dean of the Faculty of Civil Engineering, Budapest (2011) ISBN 978-963-313-042-1

International conference papers

6. **Lakatos É., Bojtár I.:** The Biomechanical Behaviour of the Trabecular Bone Surrounding Dental Implants, *Proceedings of the 3rd Hungarian Conference on Biomechanics*, Budapest, 4-5. July 2008., pp. 159-167
7. **Lakatos É., Bojtár I.:** Stochastically Generated Finite Element Beam Model of the Trabecular Bone Surrounding Dental Implants, *International Conference on Tissue Engineering*. Leiria, Portugal, 9-11 Sept. 2009., pp. 257-264

Conferences

8. **Lakatos É., Bojtár I.:** Stochastically Generated Finite Element Beam Model for Dental Research, *17th Inter-Institute Seminar for Young Researchers*, Cracow, Poland, 22-23 May 2009, pp. 9
9. **Lakatos É., Bojtár I.:** Mechanical behavior of the human jawbone: Determination of the anisotropy using micro-CT imaging *(in Hungarian)*, *XI. MAMEK,* Miskolc, Hungary, 29-31. August 2011.
10. **Lakatos É., Magyar L., Bojtár I.:** Material properties of the mandibular trabecular bone, *28th Danubia-Adria-Symposium on Advances in Experimental Mechanics*, Siófok, Hungary, 28 Sept. – 1. Oct. 2011.

Seminars

11. **Lakatos É.:** The Biomechanical Behaviour of the Human Jawbone *(in Hungarian)*, *Seminar of the BME Department of Applied Mechanics and Department of Structural Mechanics*, Budapest, Hungary, 2007
12. **Lakatos É.:** Finite element frame model of the trabecular bone microstructure – applications is the oral implantology *(in Hungarian), Seminar of the BME Department of Applied Mechanics and Department of Structural Mechanics*, Budapest, Hungary, 2010
13. **Lakatos É.:** The Biomechanical Behaviour of the Trabecular Bone Surrounding Dental Implants *(in Hungarian), X. ANSYS Conference*, Sóskút, Hungary, 21. April 2011.
14. **Lakatos É., Bojtár I.:** Bone material properties for dental surgical application *(in Hungarian), Bioinformatics and Engineering methods in Medicine Seminar,* Budapest, Hungary, 2. June 2011.
15. **Lakatos É.:** Numerical analysis of dental implants and the surrounding bone: Trabecular bone material properties *(in Hungarian), Biomechanical research in the Faculty of Civil Engineering Seminar,* Budapest, 21. June 2011.