



ISB 2013
BRAZIL

XXIV CONGRESS OF THE INTERNATIONAL
SOCIETY OF BIOMECHANICS

XV BRAZILIAN CONGRESS
OF BIOMECHANICS

COMPUTATIONAL EVALUATION OF MEDULLAR BONES MATHEMATICAL MODELS AND THEIR INTERACTION WITH DENTAL IMPLANTS.

¹Álvaro F. Moura, ²Petrus G. Nóbrega e ²Selma H. Nóbrega

¹Mechanical Engineering Post Graduation Program – PPGEM/UFRN

²Federal University of Rio Grande do Norte, Natal, RN, Brazil – e-mail: nobrega@ufrnet.br

SUMMARY

A successful dental implant treatment basically depends on good osseointegration and bone remodeling processes, both responsible for a proper support system. Changes of medullary bone tissue, observed in surroundings of implant systems, are related with their own anatomical feature, geometry, length, quantity and distance among these structures. The type and size of the prosthesis also influence the bone remodeling process. Considering the shape variation and their consistency in different regions of the arcade, the bones are classified into four groups named as Grade I to IV. It is necessary the use of a computational tool in order to analyze the stresses developed in bone tissue, influenced by all these variables, predicting a reliable biomechanical response of the bone remodeling process. Computational codes based on Finite Element Method (FEM) have been used to investigate the performance of the implant systems successfully. In this work, discussions about different mathematical models for the medullary bone tissue are presented in order to compare their response with those usually evaluated in FEM analysis.

INTRODUCTION

Dental implant systems consist of the part (“implant”) inserted into bone, the abutment and the artificial crown (“prosthesis”). This system, when well designed, ensures no failures occurrences and the appropriate transference of masticatory forces to the interfacial bone tissue. A successful dental implant treatment and its suitable support system depend on a good osseointegration and bone remodeling process. Some researchers point that two facts must be considered when studying implant systems: the nature of the masticatory forces, and the biomechanical knowledge about the bone tissue behavior.

The high complexity of the biomechanical process is related to the anatomical feature of the tissues, the implant geometry, its quantity and the distance among these structures. The type and shape of the prosthesis also influence the bone remodeling process.

In order to prevent failures occurrences and ensure the longevity of the oral rehabilitation, numerical-computational tools based on the Finite Element Methods (FEM) are used to evaluate the stresses and strains distributions in surroundings of implant structures. In last decades, FEM has

been successfully used in many Dentistry and others Health areas. One important step for FEM analysis is to establish a reliable mathematical model, including the hypothesis of kinematics, loadings, boundary conditions, geometries and others.

However, mathematical models that consider all real complexities are not viable and simplifications are unavoidable. In implant systems, in spite of the diversity of shape and consistency of the bones tissues in arcades (maxilla and mandible), they are usually modeled as one single material with specific mechanical properties.

In this paper, numerical results of stresses developed in the neighborhood of implant structures are presented. The models consider different shape and densities of bones tissues present in the human arcades, which are classified in four groups: Grades I to IV. Figure 1 shows these four groups, where the dense superficial bone (cortical) is in yellow and the porous internal bone (medullar) is in red.

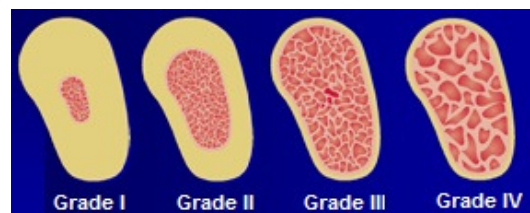


Figure 1: Bone classification (Lekholm and Zarb, 1985).

METHODS

ADINA 8.8 code was used for FEM analysis. Figure 2 shows the computerized tomography (CT) scan of the bones classified as Grades II and IV, respectively. The finite element meshes of these bones, including an implant structure can be seen in Figure 3.



Figure 2: CT images of bones Grades II e IV.

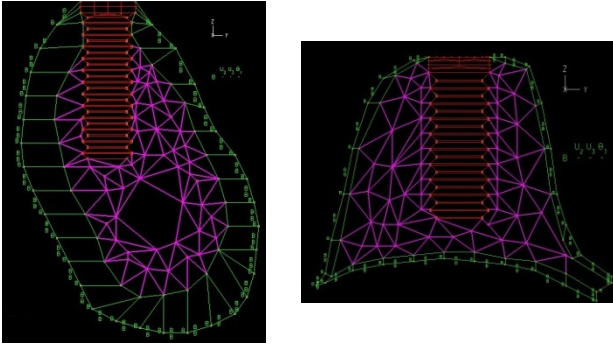


Figure 3: Finite elements meshes.

For cortical bone (green lines) and implant (red lines) 2D Plane Strain elements (9 nodes) were used. For the medullar bone (magenta lines) the simulations were performed with 2D Plane Strain elements (9 nodes) or truss elements (2 nodes). All linear and angular displacements around the bone structure were fixed. An uniform vertical distributed pressure (equivalent to 175N concentrated force, related to a medium value of masticatory effect) is applied on top. The mechanical properties are presented in Table 1 and are based on the values published by two different papers [1,2].

Table 1: Mechanical properties

Materials	Young's modulus (GPa) [1]	Young's modulus (GPa) [2]	Poisson's ratio
Titanium	110,0	103,0	0,33
Cortical bone	13,7	10,0	0,30
Medullar bone	1,37	0,25	0,30

Four mathematical models were built for the medullar tissue: MODEL1 = 2D elements and properties of medullar bone [1]; MODEL2 = 2D elements and properties of medullar bone [2]; MODEL3 = truss elements and properties of medullar bone [1]; and MODEL4 = truss elements and properties of cortical bone [1].

RESULTS AND DISCUSSION

Numerical results of stresses intensities, shown in Table 2, are normalized in relation to the response of Model1, usually found in papers.

Table 2 – Max. effective stresses – implant/cortical region.

Bone	Model 1	Model 2	Model 3	Model 4
Grade II	1,0000	1,2078	1,0398	0,8325
Grade IV	1,0000	1,0872	1,0584	0,8238

The stresses values for Model 2 are always bigger than for those of Model 1. Obviously this fact is related to the Young's Modulus presented in [2], lower than [1], according Table 1.

It is important to remember that the materials are human tissues and bones, making difficult the evaluation of the mechanical properties. The values based on [2] and indicated in Table 1 are medium values, obtained by

empirical relationships between Young's modulus and apparent bone density.

Moreover, the medullar bone is a porous media, and the usual mathematical models are based on the Continuum Mechanics Theory. This means a limitation between the reality and numerical simulations. Model 3 and 4 with truss elements are proposed in order to evaluate this non continuum media. The truss elements would be set where real bones segments could take place, preserving all the intermediate regions of the soft tissue.

Stresses distributions on the cortical and medullar regions are presented in Figure 4.

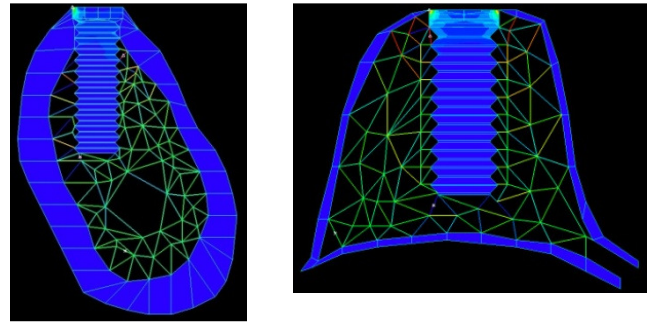


Figure 4: Effective stresses on bones Grade II and IV.

It is important to point out the complexity to define the truss mesh density. Models 3 and 4 analyzed with 30% more elements, and the same Young's modulus, produced a result approximately 5% lower, what is coherent with a stiffer model.

CONCLUSIONS

The usual FEM model deals with 2D Plane Strain elements to simulate the medullar bone microstructure which is, in fact, composed by bone segments and empty spaces (fluids, fat cells, nerves and others). Therefore, a carefully definition of material properties is required, what is something difficult to acquire. In this work, models were discussed in order to get more realistic results: mechanical properties based on empirical formulas and simulation of the medullar bone as a bone network. In future, new models will be also proposed based on CT images to determine a better procedure to simulate the medullar tissue.

REFERENCES

1. Lin, D. et al. Dental implant induced bone remodeling and associated algorithms. *Journal of the Mechanical Behavior of Biomedical Materials*, v. 2, n. 5, p. 410-432, 2009.
2. Wang, J. C.; Huang, S. F; Lin, C. L. Biomechanical responses of endodontically treated tooth implant-supported prosthesis. *Journal of Endodontics*, 36(10), 1688-1692, 2010.