MODELING OF BONE STRUCTURES THROUGH FINITE ELEMENT METHOD

Cláudio Luís Hayasaki
Department of Mechanical Engineering - Faculty of Engineering - State University of São Paulo (Unesp)
Av. Luiz Edmundo Carrijo Coube s/n, Bauru, São Paulo, Brazil
claudiolha@yahoo.com

Edson A. Capello Sousa
Department of Mechanical Engineering - Faculty of Engineering - State University of São Paulo (Unesp)
Av. Luiz Edmundo Carrijo Coube s/n, Bauru, São Paulo, Brazil
capello@feb.unesp.br

Abstract. *The objective of this paper was to simulate through the Finite Element Method, the bone structure of rabbits, particularly the tibia, when subjected to a determined static transversal loading. The mechanical behaviour of biomechanical structures, as tibia, femur, bones in general, are too much similar as mechanical structures, for example beams, bars, pipes, bridges, trusses, and so on. There were several difficulties in generating the three dimensional model, we can numerate them. One of them was to build the model exactly with all the irregular surfaces. For solving this problem, we measured along the length of the tibia for each five millimeters, beginning on the position of the implant. The second difficulty was to separate different materials that constitute the body of the bone, for instance, the cortical bone and the medullar bone, as well as the region of implant of titanium. We differentiate the regions where each type of bone occurs, including the osseointegrated region. The third difficulty was to represent the load that acted over the bone structure. We adopt the load as a distributed pressure located over the region of the implant. The finite element model was simulated by Ansys R10, where was applied tetrahedrical elements for generating the mesh of the model. For validating the model, the results were checked with the experimental results. We compare the results of intensity of maximal stress generated by the model with the maximal stress obtained experimentally. We conclude that was a reasonable approach in both cases: osseointegrated and non-osseointegrated implant, considering the accuracy of the constructed model. It was noticed that the osseointegrated region presented a higher mechanical stiffness than the case where the implant is put without the osseointegration. This feature was also verified in the experimental results.*

Keywords*: finite element, implant, osseointegration, simulation, modeling*

1. INTRODUCTION

Researches involving engineering and medicine have been developing too fast in the last decades. The evolution of the computational systems, as well as the pertinent software contributed to the interface between these areas.

The simulation by Finite Elements has been performed when involves mainly the use of implants in bone regions, like human mandible and bones, organs and until the skull was an object of study.

The application of the Finite Element Method has been a tool of engineering with higher potential in biomechanical applications, mainly where the invasive method to insertion of implants inaccessible or improbable to be performed.

In principle the application of the Finite Element Method for solving structural problems in prosthesis is quite appropriate, because the analysis of a prosthesis structure can be developed in analogous form to performed evaluation in stress analysis of mechanical structures.

The models of structures can be two or three-dimensional. The efficiency and the proximity of the real physical conditions are some of the advantages of the three-dimensional models, which by this, passed to be more used.

In the development of biomechanical products, it should have in mind the importance of simulation and modeling of these products, because permit the evaluation of structural behaviour of these prostheses when substitute the function of a part of the organism. The modeling and simulation permit the validation of the proposed element, permitting a possible optimization in the design of the product (prosthesis) to be fabricated.

The evaluation and development of biomechanical products need of structural analysis and modeling to evaluate its mechanical strength, mainly when subjected to external forces. In this paper is done the validation of the model of a bone structure, where it can verify the structural behaviour of a bone that received a surgical implant. Thus, it has verified the ability of computational model of Finite Element, in presents the biomechanical behaviour of bone structure. A rabbit tibia was used to present the bone structure, and the structural behaviour of bone was verified, when this is subjected to a transversal load.

2. BIBLIOGRAPHIC REVISION

According to the context of application of Finite Element Method in the filed of the biomechanics, it was revised the bibliography that deal with these modeling of bone structures.

BRANEMARK (1983), published article with a short revision of many investigations that lead to the clinical application of osseointegrated implants of titanium. It described the first experiences for study of osseous reparation after breakings utilizing optical chambers by transilumination of small areas in fibulas of rabbits. The rigid incorporation of bone over the titanium metal leaded the author to suggest the use of this material for fixation of osseous autogen enxertos and the development of implants in shape of screw to rehabilitation of rebordos desdentados.

MATTILA (2000) did a research to clarify if a diet of xylitol would affect the resorption, structure and the biomechanical osseous properties of saudaveis rats and if the same diet could offer some prevention against the rise of resorption, reduction of trabeculado and biomechanical weakness of bone rats with osteoporosis induced experimentally. The author verified a reduction of osseous resorption in male saudáveis rats of 3 months and protected of manner statistically significant against the rise of resorption in female ovariectomizadas rats of 3 months, according to the measure of urinary excreption of H^3 , after the administration of H^3 – tetraciclina pré-marcada. A drawing with the sketch of 3 points bending test, extracted of article can be viewed in Figure 1. The author concluded that the results reinforced the hypothesis of that the oral administration of xylitol protects against the progression of experimental osteoporosis. Such diet was effective as in the increment of osseous mass in male rats, as prevented osseous losses in female rats submitted to the withdrawal of ovario, like a interesting alternative in research of new options of prevention of this disease.

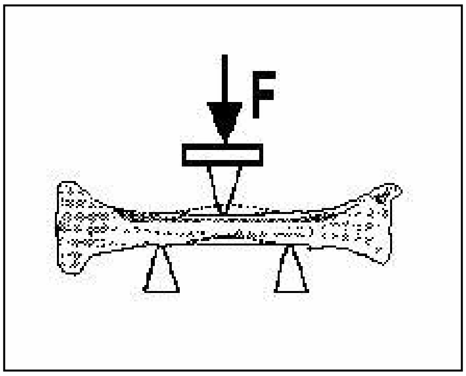

Figure 1. Drawing of MATTILA (2000), presenting the bending test by 3 points.

Utilizing the Finite Element Method, three-dimensional models were generated and applied in many situations like for example in segments of spine using computerized tomography, KUMARESAN et al (1997), in skeleton analysis, analysis and design of orthopedic devices and analysis of rise of tissue, PRENDERGAST (1997), in the model of human femur after artroplasty of hip joint utilizing tomography, ZANNONI et al (1998), in the generation of mesh in structural analysis of teeth, LIN et al (1999), in the generation of three-dimensional model of the articular disc of human temporomandibular joint done by BEEK et al (2000).

Another applications of Finite Element Method using models of rupture of bones, where were examined the performance of 9 theories based in stress and strain done by KEYAK and ROSSI (2000), or in the generation of model of mesh of a skull of just been born with the objective of studying its deformation during its gestation, according to LAPEER and PRAGER (2000), or in the generation of bi-linear model that was applied for simulation of failure in bovine tibias according to NIEBUR et al (2000). In the Figure 2, is presented the skull modeled by LAPEER and PRAGER.

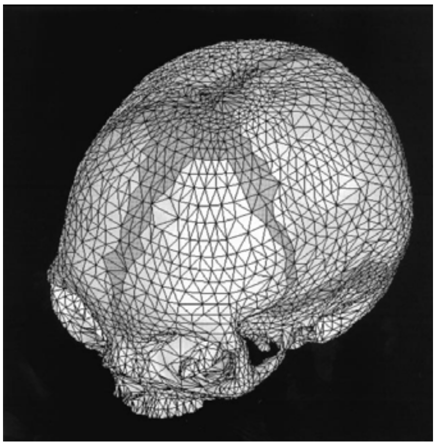

Figure 2 – Skull modeled by LAPEER and PRAGER (2000)

The Figure 3 shows a three-dimensional model of Finite Elements of proximal tibia with a triangular implant, according to Simon (2003).

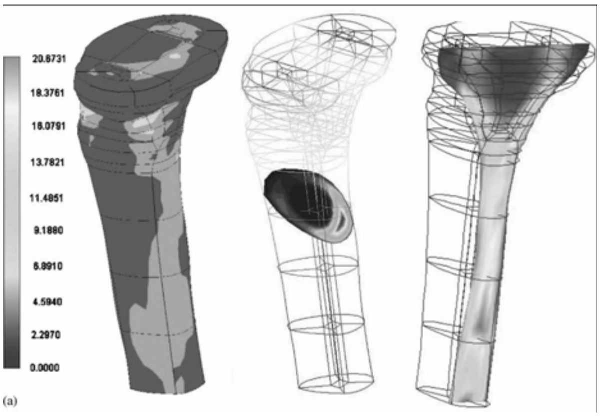

Figure 3. Tibia with a triangular implant

The Figure 4 shows stress analysis for a model of a tibia, according to Muller-Kärger (2004).

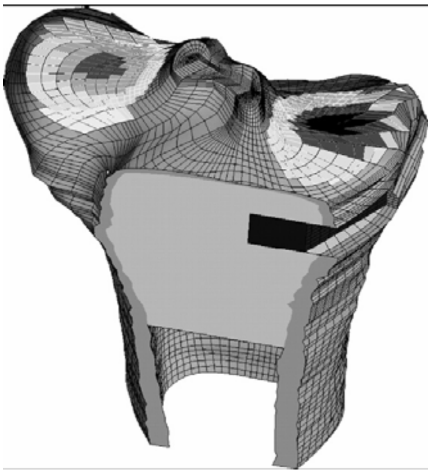

Figure 4. Von Mises stress for a model of a tibia.

Combination of high precision in kinematics in vivo three-dimensional and of the model of inferior member based in explicit techniques of Finite Elements was study of BEILLAS et al (2004). In Figure 5, is presented the model of internal structures.

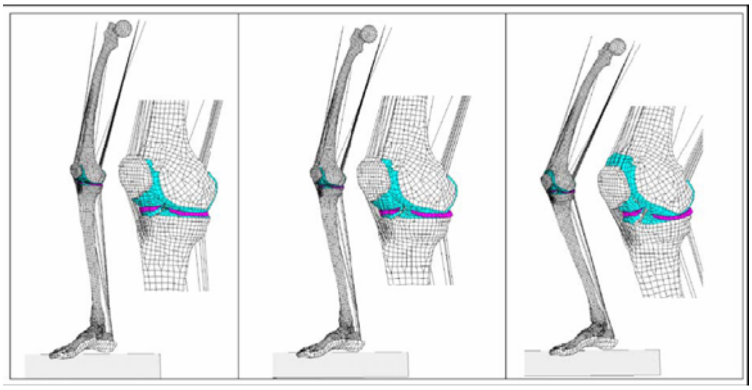

Figure 5. Models of internal structures of joints done by BEILLAS(2004).

3. MATERIAL AND METHODS

In this paper it was used the software Ansys Release 10.

To solve the problem of the irregularity of bone structure, it began from the implant (zero position) and measured for each 5 millimeters along the length of the bone. Thus, it approximated the points of a same plane to form an transversal area. It bonded 2 consecutive areas by a command of the software Ansys. The first area bonded with the second, the second bonded with the third and so on. Therefore, the volume was generated by arbitrary areas defined by lines. The total length of the bone was approximately 12 centimeters.

Figure 6. The bone structure with the implant.

The Figure 7 shows the selection of materials for the bone structure in the Group Osseointegrated. There are cortical bone (green), medullar bone (blue), implant (red) and the region of osseointegration (green-sphere).

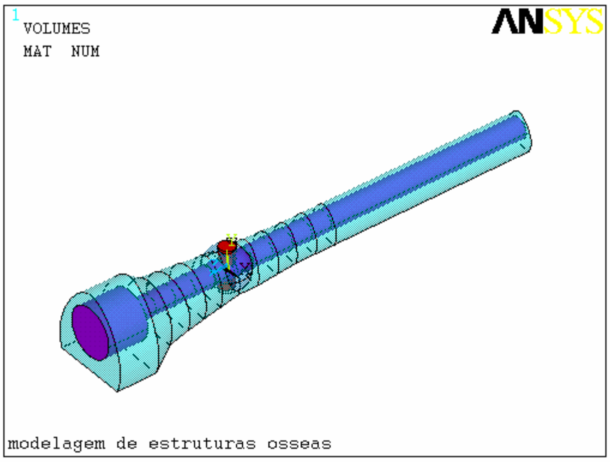

Figure 7. The different materials in bone structure.

The region of osseointegration is detailed in the Figure 8.

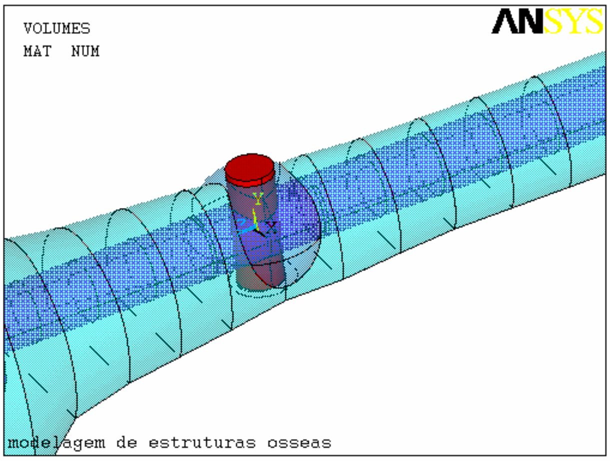

Figure 8. The region of osseointegration.

The Table below shows the material properties applied in the numerical model.

Table 1. Material properties.

Type of bone	Young’s modulus	Poisson ratio
Cortical bone	20 GPa	0.3
Medular bone	5 GPa	0.3
Implant	110 GPa	0.3
Osseointegration	20 GPa	0.3

The Figure 9 shows the element SOLID 92. It has 10 nodes and each node has 3 degrees of freedom.

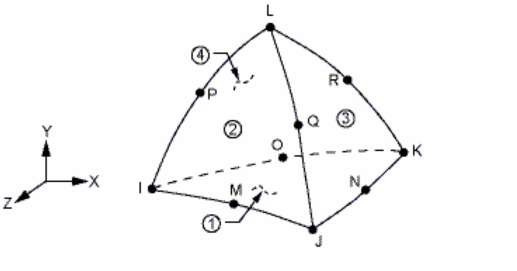

Figure 9. Element Solid 92 of Ansys

The Figures 10 and 11 shows the free mesh for the Group Non-osseointegrated and Group Osseointegrated respectively.

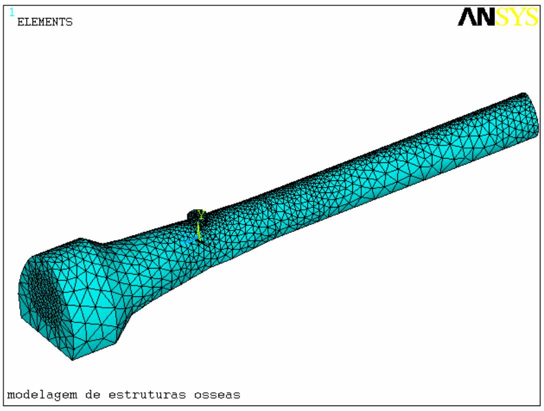

Figure 10. Mesh for non-osseointegrated group

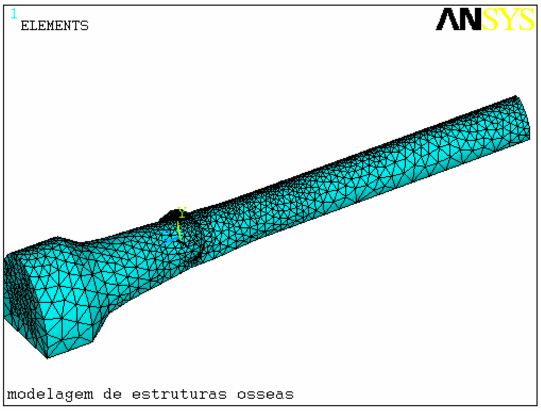

Figure 11.Mesh for osseointegrated group

The Figures 12 and 13 shows the boundary conditions and loads for the Group Non-osseointegrated and Group Osseointegrated.

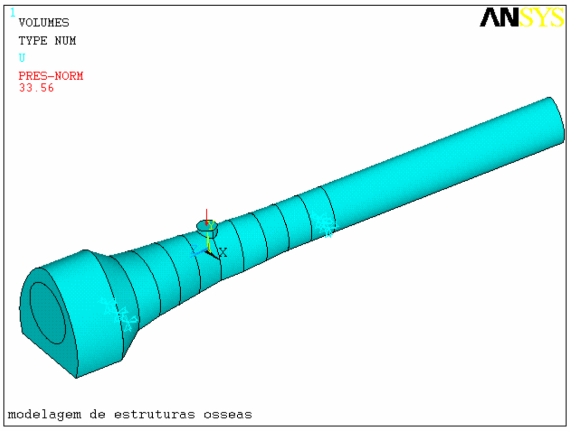

Figure 12. The load and restrictions in Group Non-osseointegrated.

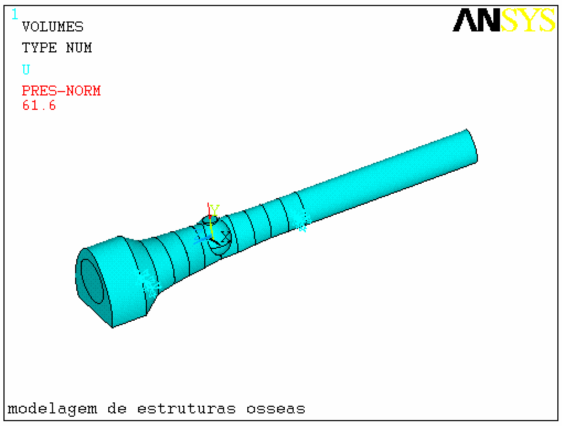

Figure 13. The load and restrictions in Group Osseointegrated.

4. RESULTS

According to Table 1, in the Non-osseointegrated Group there was a considerable decrease of mechanical strength, 43.9 % in average. Varying the conditions alternatingly for the left and right tibia according to the position of implant and measuring the found load exactly in the instant of fracture, the Table 2 was done. Notice that the average load of fracture was 37.06 Kgf with implant non-osseointegrated, while the tibia without implant presented an average load of 66.03 Kgf.

Table 2. Experimental data for the Non-osseointegrated Group
Observation: E= left D= right, sem = without implant, com= with implant..

Coelho	Peso	Condição	Carga- em kgf	Decréscimo de Resistencia (%)
1	4	sem E/ com D	68,20/36,40	-46,62
2	3,8	sem D/com E	65,75/34,20	-47,98
3	4,3	sem E/com D	61,40/41,95	-31,67
4	4	sem D/com E	60,25/39,80	-33,94
5	4,3	sem E/com D	67,10/44,30	-33,97
6	3,8	sem D/com E	70,10/29,10	-58,48
7	4,4	sem E/com D	50,65/39,65	-21,71
8	4,2	sem D/com E	69,75/38,30	-45,08
9	4,1	sem E/com D	85,95/35,55	-58,63
10	3,8	sem D/com E	61,15/31,40	-48,65
Medias	4,07	sem/ com	66,03/**37,06**	**-43,87**
desvio padrão	0,23		9,12/4,69	

Table 3. Experimental data for the Osseointegrated Group

Coelho	Peso	Condição	Carga - em kgf	Acréscimo de Resistência (%)
11	4,2	sem E/ com D	64,05 /71,45	11,55
12	4	sem D/ com E	56,70 /83,25	46,82
13	4,2	sem E/ com D	64,95 /73,55	13,82
14	4,3	sem D/ com E	52,75 /58,60	11,09
15	4,1	sem E/ com D	54,75 /64,65	18,08
16	3,9	sem D/ com E	34,10 /53,35	56,54
17	3,8	sem E/ com D	58,6 /62,75	7,08
18	4,3	sem D/ com E	54,20 /71,05	31,08
19	4,5	sem E/ com D	50,30 /67,85	34,89
20	3,8	sem D/ com E	57,20 /73,40	28,32
médias	4,11	sem/com	54,76 /**67,99**	**24,16**
desvio padrão	0,23		8,60/ 8,53	

The Figure 14 shows the intensity of stress in the osseointegrated model, while the Figure 15 presents the Von Mises stress in the osseointegrated region of a detailed view.

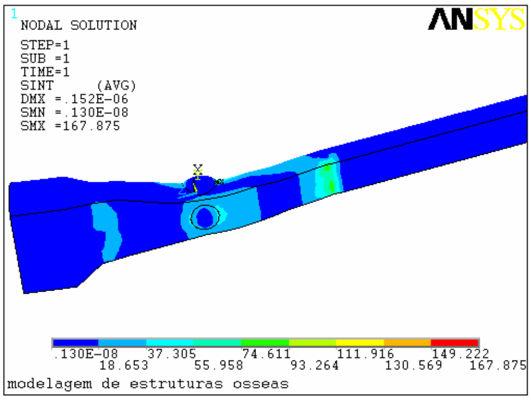

Figure 14. Stress in the osseointegrated model

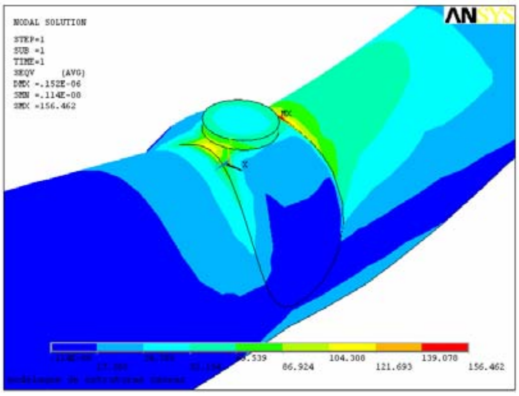

Figure 15. Stress in the osseointegrated region

In the Figures 16 and 17, it can make a comparison between the intensity of stress for the cases Osseointegrated and Non-Osseointegrated. It notice that due to osseointegration, which caused a increase in mechanical strength, the level of stress in the Osseointegrated Group is lower that the Non-osseointegrated Group.

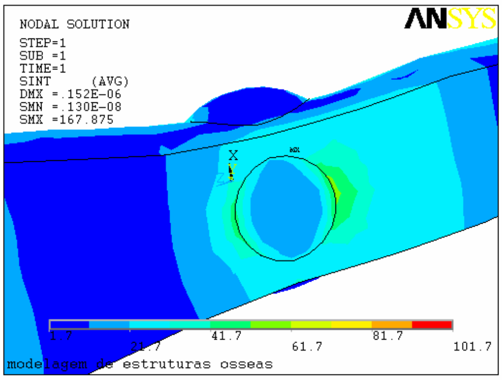

Figure 16. Stress analysis for the GO group

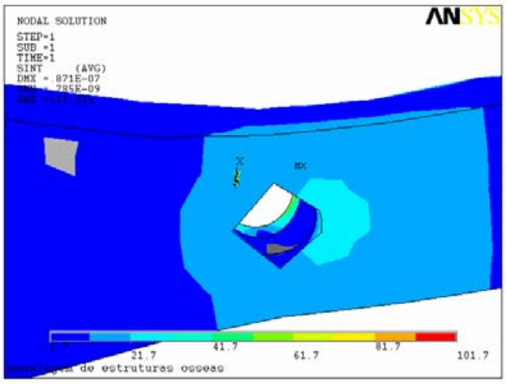

Figure 17. Stress analysis for the GNO group

The Figures 18, 19 and 20 shows the Von Mises stress analysis in the complete structure for the three cases. (GSI means Group without implant)

The stress level near the implant in the GSI group is close to 45 MPa, in the GNO group is close to 52 MPa, but in GO group is close to 34 MPa. This decrease of stress in GO group is due to the osseointegration, which increase the mechanical strength and reduces the stress in the region.

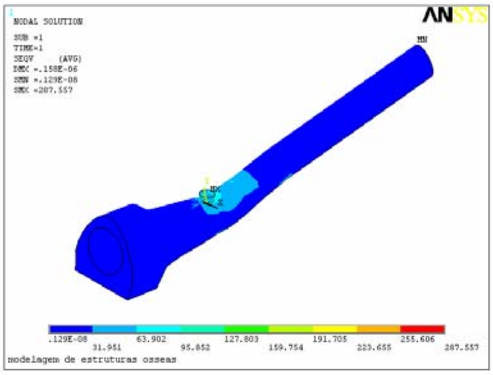

Figure 18. Stress analysis for GSI group

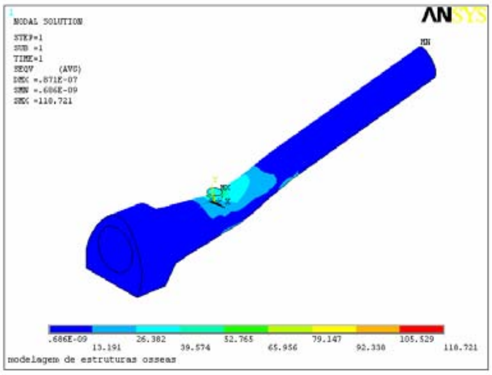

Figure 19. Stress analysis for GNO group

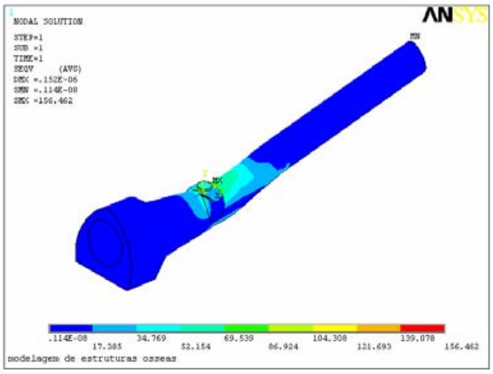

Figure 20. Stress analysis for GO group

5. CONCLUSION

It was verified that there was simplifications in the modeling of bone structures. Normally, the bone is non-linear and anisotropic. In this paper, it was assumed that the bone (tibia) was linear, isotropic and elastic.

The region of osseointegration yielded an additional strength to the bone structure, that caused a decrease in stress levels in the region of the calcification. In the non-osseointegrated structure it occurred the inverse, since there was a decrease of mechanical strength, and there was an increase in stress levels.

For the 3 analyzed cases, Group without implant, Group non-osseointegrated and Group Osseointegrated, the found Von Mises stress near the region of implant were higher than the found values in the literature (fracture stress) for the bone structure.

As future papers, it is suggested the inclusion of real material properties, the improvement in the construction of geometric model and the inclusion of dynamic forces in modeling of bone structures.

6. ACKNOWLEDGEMENTS

I would like to thank to CAPES, by the financial support and the professor Edson A. Capello Sousa, by the fundamental help in the elaboration of this paper.

7. REFERENCES

Beek et al, 2000, “Three-dimensional finite element analysis of the human temporomandibular joint disc”, Journal of Biomechanics, p.307-316, v.33.

Beillas et al, 2004, “A new method to investigate in vivo knee behaviour using a finite element model of the lower limb”, Journal of Biomechanics, p.1019-1030, v.37.

Branemark, P. I. , 1983, “Osseointegration and its experimental background”, J.Prosth. Dent., p.399-410, v 50.

Dekon, A . F., 2004, Doctor Thesis, “Resistência à fratura por flexão de tíbias de coelho hígidas e transfixadas por implantes osseointegráveis”, Bauru, Brazil.

Keyak and Rossi, 2000, “Prediction of femoral fracture load using finite element models: an examination of stress and strain based failure theories”, Journal of Biomechanics, p. 209-214, v.33.

Kumaresan et al, 1997, “Finite element biomechanics of cervical spine interbody fusion”, Proceedings- 19th International Conference IEEE/EMBS 30 Oct-2 Nov, Chicago USA.

Lapeer and Prager, 2000, “3D Shape recovery of a newborn skull using thin-plate splines”, Computerized Medical Imaging and Graphics, p. 193-204, v.24.

Lin et al, 1999, “Automatic finite element mesh generation for maxillary second premolar”, Computer Methods and Programs in Biomedicine, p. 55-64, v. 72.

Mattila, P. , 2000, “Dietary xylitol in the prevention of experimental osteoporosis. Beneficial effects on bone resorption, structure and biomechanics”, Net, February 2000, http://herkules.oulu.fi/isbn951425158X/html/

Müller-Kärger et al, 2004, “P-version of the finite element method for highly heterogeneous simulation of human bone”, Finite Elements in Analysis and Design, p. 757-770, v.40.

Niebur G.L. et al, 2000, “High resolution finite element models with tissue strength asymmetry accurately predict failure of trabecular bone”, Journal of Biomechanics, p. 1575-1583, v.33.

Prendergast P. J. et al, 1997, “Finite element models in tissue mechanics and orthopaedic implant design”, Clinical Biomechanics, p. 343-366, v. 12.

Simon, U. et al, 2003, “Influence of the stiffness of bone defect implants on the mechanical conditions at the interface-a finite element analysis with contact”, Journal of Biomechanics p. 1079-1086, v.36.

Zannoni, C. et al 1998, “Analysis of titanium induced CT artifacts in the development of biomechanical finite element models”, Medical Engineering and Physics, p.653-659, v. 20.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.